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Utilization of model-based definition within mechanical design and manufacturing

Master's Thesis

Thesis submitted for examination for the degree of Master of Science in Technology.

Hyvinkää 4.4.2016

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Abstract of master's thesis

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Abstract

Model-based definition is a design method in which all the product and manufacturing information of a part is included in the 3D-model, instead of a conventional 2D-drawing. The objective of this thesis was to determine the possibility to implement model-based definition to the currently used systems of a target company. Another target was to determine how the data transfer between various organizations could be conducted without the 2D-drawings. In addition, the utilization of the 3D-models within manufacturing processes and the possible benefits within the entire process considering time savings and quality improvements were investigated.

Several sources indicate that model-based definition offers various advantages over the conventional 2D-definitions. The benefits include time savings within design, reduction of misunderstandings in the design documents, and the extraction of tolerances and annotations of a 3D-model in the manufacturing processes. To verify the claimed benefits, several studies were conducted. The studies were related to the design processes, design items, manufacturing processes and implementation of model-based definition in two CAD-software.

The CAD-software used in the target company is capable of generating fully defined 3D-models with no additional investments. Regarding the manufacturing processes investigated within this thesis, it can be stated that almost every manufacturer is capable of utilizing 3D-models. However, there are various software used among the manufacturers, for which so-called neutral formats need to be used. By using the neutral formats, parts of the intelligence within the models disappear and the reuse possibility of the model properties becomes more difficult.

In conclusion, including all the product and manufacturing information to the 3D-model makes the product definition unambiguous and eliminates the possibility of differences between the 3D-model and the 2D-drawing. In addition, the geometry is generally easier to understand in a 3D-format. However, no time savings in the design phase could be achieved in the time studies conducted during the thesis. Certain time saving could most likely be achieved if the design tools of model-based definition were more sophisticated and more familiar to the designers. Due to several software among the manufacturers, further studies on how the fully defined 3D-models and neutral file formats can be utilized within the software, should be conducted. Hence, the utilization of model-based definition should be tested in a pilot project.

Keywords MBD, MBE, PMI, 3D-model, 2D-drawing, CAD, CAM, mechanical design, manufacturing

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Tiivistelmä

Mallipohjainen määrittely tarkoittaa kappaleen tuote- ja valmistustietojen määrittelemistä 3D-mallissa perinteisen 2D-piirustuksen sijaan. Tämän diplomityön tarkoitus oli selvittää kuinka mallipohjainen määrittely voidaan toteuttaa kohdeyrityksen nykyisillä järjestelmillä, sekä miten tiedonsiirto käytännössä toteutettaisiin ilman 2D-piirustusta. Näiden lisäksi selvityksen alla olivat kuinka 3D-malleja voidaan hyödyntää nykyisissä valmistusympäristöissä ja voidaanko mallipohjaisen määrittelyn avulla saavuttaa etuja kokonaisprosessissa niin ajansäästön kuin laadunparannusten osalta.

Mallipohjaisen määrittelyn eduiksi mainitaan tyypillisesti lyhentyneet suunnitteluajat, vähentyneet väärinkäsitykset suunnitteludokumenttien ymmärtämisessä sekä mallissa olevien mittojen ja toleranssien hyödyntäminen valmistusprosesseissa. Jotta kyseiset edut voitaisiin todentaa, toteutettiin työn aikana useita tutkimuksia. Tutkimukset liittyivät suunnitteluprosesseihin, suunniteltuihin osiin, valmistusprosesseihin sekä mallipohjaisen määrittelyn toteuttamistapoihin CAD-ohjelmistoissa.

Yrityksessä käytössä olevalla suunnitteluohjelmalla pystytään luomaan täysin määriteltyjä 3D-malleja ilman ylimääräisiä investointeja. Tutkimuksissa selvitettyjen valmistusprosessien osalta voidaan todeta, että 3D-malleja pystytään myös hyödyntämään lähes jokaisen valmistajan prosesseissa. Prosesseissa käytetyt ohjelmistot kuitenkin vaihtelevat, minkä vuoksi tiedonsiirtoon joudutaan usein käyttämään niin sanottuja neutraaleja tiedostoformaatteja. Näitä käytettäessä osa mallissa olevista ominaisuuksista häviää ja mallissa olevien toleranssien ja muiden merkintöjen hyödyntäminen hankaloituu.

Kaikkien tuote- ja valmistustietojen sisällyttäminen 3D-malliin tekee määrittelystä yksiselitteistä ja poistaa ristiriidan mahdollisuuden mallin ja piirustuksen väliltä. Lisäksi 3D-muodossa oleva tieto on helpompi ymmärtää. Työssä tehtyjen mittausten perusteella ajansäästöä ei kuitenkaan saavutettu suunnitteluvaiheessa. Ajansäästöjen saavuttaminen olisi kuitenkin todennäköistä, jos mallipohjaisen määrittelyn työkalut olisivat kehittyneempiä sekä tutumpia suunnittelijoille. Nykyisessä suunnittelu- ja valmistusympäristössä käytetään lukuisia eri järjestelmiä, mistä johtuen järjestelmien yhteensopivuus määriteltyjen 3D-mallien osalta on kokonaisuuden kannalta erityisen tärkeää. Järjestelmien yhteensopivuutta ja neutraalien formaattien hyödyntämistä tuleekin jatkossa selvittää lisää. Tämän vuoksi mallipohjaista määrittelyä tulisikin kokeilla pilottiprojektin muodossa.

Avainsanat MBD, MBE, PMI, 3D-malli, 2D-piirustus, CAD, CAM, mekaniikkasuunnittelu, valmistus

Preface

The creation of the Master's Thesis has been a challenging, but rewarding project. I have learned many new things about the company, and about the whole design and manufacturing environment. Due to all the knowledge I have gathered, I am now more prepared for the new challenges.

I would like to thank my advisor Mats for the great advices and feedback as well as for the possibility to concentrate fully on the thesis. Compliments also belong to my colleagues who have helped me to create several studies, given me excellent contacts and helped in the proofreading. I would also like to thank my supervisor Professor Kari Tammi for the constructive feedback especially concerning the scientific nature and a proper structure of the thesis.

Special thanks belong to my parents who have always supported my decisions and emphasized the significance of a good education. The greatest thanks belong to my dear girlfriend Riikka for the help in the proofreading and for the support she has given me during the years.

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Matti Pajula

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Abbreviations

2D	Two Dimensional
3D	Three Dimensional
ASCII	American Standard Code for Information Interchange
ASME	American society of mechanical engineers
ASTM	American society for testing and materials
BOM	Bill of materials
CAD	Computer aided design
CAM	Computer aided manufacturing
CIM	Computer integrated manufacturing
CMM	Coordinate Measurement Machine
CNC	Computerized numerical control
CTO	Configured to order
ETO	Engineered to order
GD&T	Geometric dimensioning and tolerancing
ISO	International Organization for Standardization
MBD	Model-based definition
MBE	Model-based enterprise/engineering
NC	Numerical control
PDM	Product data management
PLM	Product lifetime management
PMI	Product and manufacturing information
SFS	Standardization Organization of Finland
TDP	Technical data package

1 Introduction

Increased productivity and decreased costs are the current trends within the industry. Those factors should thus be taken into account in all the stages of the product lifecycle including product development and manufacturing. Process improvements might include optimization of the methods and elimination of the overlapping tasks. The department, for which this master's thesis is made, utilizes 3D-CAD for modeling. However, the final product of the design, which is delivered to the manufacturing, is usually a 2D-drawing of a part or an assembly.

An approach that has been claimed to make the product development process more efficient and improve the utilization of 3D-CAD software and the 3D-models, is called model-based definition (MBD). MBD is an approach to the design, in which all the product and manufacturing information (PMI) is applied as annotations to the 3D-model. This annotated 3D-model can later be utilized in all the different departments associated to the product lifetime, including the design, manufacturing and quality control. Hence, the 3D-model must include all the information that is currently expressed in a 2D-drawing to be able to perform the same tasks based on only a 3D-model. (Lubell, et al., 2012)

This master's thesis is made for Konecranes. Konecranes is the world-leading company in providing lifting solutions for various industries. Konecranes also provides service for cranes and machine tools of every manufacturer. Total group sales in 2014 was 2011 million euros. Konecranes has about 12000 employees in 48 different countries and in 600 different locations. (Konecranes, 2016)

1.1 Motivation

The 3D-CAD software were originally introduced with the argument that they would improve the efficiency of the overall process. However, a common opinion among mechanical designers is that the properties and possibilities of the CAD-software and the 3D-models are not fully exploited yet. As stated in the previous section, the current method is to generate a 2D-drawing from a 3D-model, which includes a complete definition of the product geometry. It can be assumed that the 2D-drawing is not the most illustrative way to represent the product geometry and annotations. Understanding the product geometry based on only the 2D-representation may require a large amount of time and possibly lead to misunderstandings. An additional fact is that the capability of engineers to understand and produce 2D-drawings is becoming weaker, as the majority of educational institutes focus on teaching 3D-modeling skills instead of 2D-drawings.

In addition to better product visualization, there are several other issues that promote the utilization of model-based definition. Currently, the annotation work is partially done twice, as the geometry and dimensions are already defined in the model and later annotated to the 2D-drawing. Another issue is that the product information is currently located in two separate files: geometry in the 3D-model and annotations and projected views in the 2D-drawing. This approach might lead to conflicts and updating problems if the link between the two files cannot be established. A significant fact of making the implementation of model-based definition easier is that the majority of the CAD-software already include features that enable the 3D-annotations. In addition, several standards define how MBD should be implemented. A future trend is that the manufacturing processes are going towards digital manufacturing, and the processes can be simulated before the actual implementation. Hence, it is important that the model of the part contains the complete definition required in the different stages of the overall process.

1.2 Objective

The main objective of this thesis is to determine whether the overall process of the mechanical design and manufacturing can be made more efficient by applying the annotations straight to the 3D-model instead of applying them to the 2D-drawing. The overall efficiency depends on both the design and manufacturing processes meaning that the both processes must be evaluated.

The objective of the thesis can be expressed in the form of research questions. The questions are listed below:

- Can model-based definition be implemented in the current design and manufacturing environment?
- How can the data transfer be implemented if the 2D-drawings no longer exist?
- How can the annotated 3D-models be utilized in different stages of the process?
- Can model-based definition save time and increase quality during the overall process?

1.3 Scope

The main concept behind the scope of the thesis is that it should provide enough information to answer the research questions. On the other hand, the work should be limited so that it does not exceed the appropriate content of a master's thesis.

The field of the current industries is so wide that the needed manufacturing information depends significantly on the particular part to be manufactured. In addition, the information is represented differently depending on the industry. According to Hasari & Salonen (2006) there are several types of technical drawings. Figure 1 presents the division of the drawing types. The relevant drawing type in the scope of this thesis is the mechanical drawing. From the four types of the mechanical drawings, the relevant types here are the detail drawings, subassembly drawings and assembly drawings.

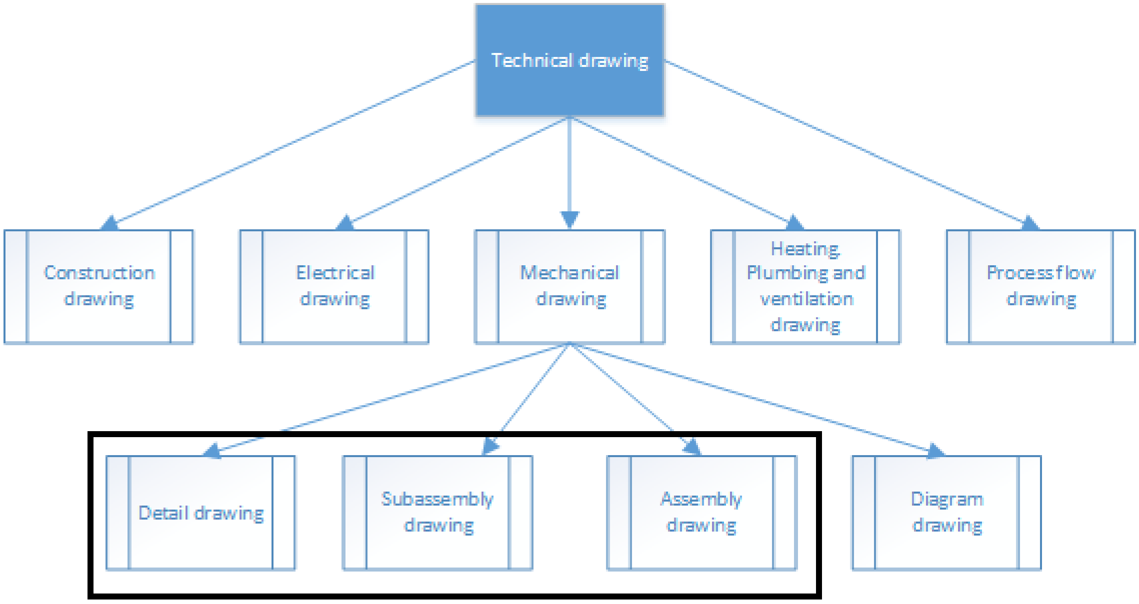


Figure 1 Division of drawing types

From the manufacturing point of view, this thesis concentrates on the typical manufacturing methods used in the metal industry. Basically, the issue to be investigated, is how the contents of the three previously mentioned drawing types can be represented in the 3D-models. In addition, the utilization of the fully defined 3D-models is investigated within the most typical manufacturing methods. The greatest importance is in the contents of the detail drawing, because that content is used for manufacturing a part.

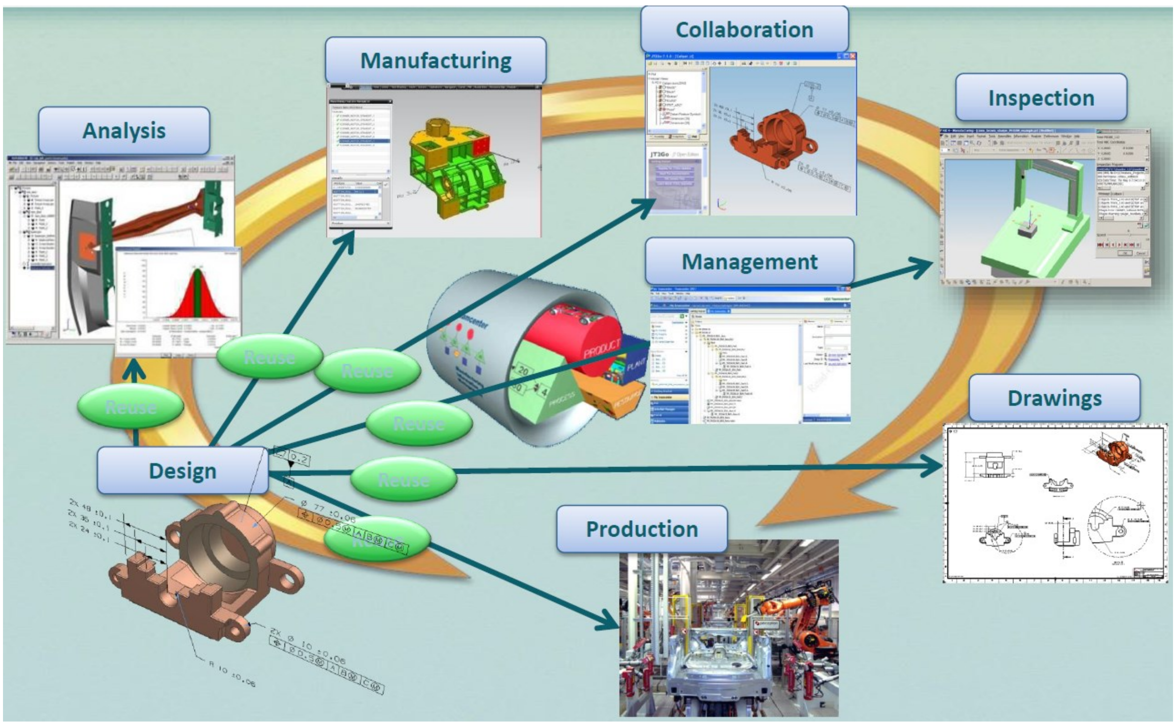


Figure 2 Utilization possibilities of an annotated 3D-model (Simons, 2016)

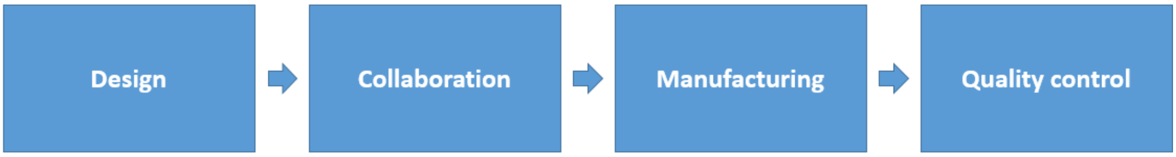


Figure 3 Process covered in the thesis

Figure 2 shows the different processes in which the annotated 3D-models can be utilized, and Figure 3 shows the sections covered within this thesis. The collaboration explains practically the method with which the data exchange between the design and the manufacturing departments is performed.

1.4 Contents

The theoretical background of the thesis concentrates on the evolution and future trends of both the mechanical design and the manufacturing. Another theory section concentrates on model-based definition in a more detailed way by providing a literature review on sections including MBD characteristics, MBD user experiences as well as MBD related standards and file formats.

To determine whether the MBD design is feasible in the current design and manufacturing environment it is important to know how the processes are implemented. In order to gather the knowledge, the manufacturing processes of certain Konecranes factories and certain subcontractors are investigated. The design process followed in the mechanical design department is also clarified. The investigations are done based on both written material as well as surveys and interviews conducted during the thesis work.

The fifth chapter concentrates on the practical implementation of MBD. The section includes investigations on how the 3D-annotation work can be performed in CAD-software and how the information can be delivered to manufacturing departments. The utilization of the 3D-models and 3D-annotations within manufacturing and quality control is also investigated. The investigations include sections such as 3D-model utilization in computer aided manufacturing (CAM) and 3D-model utilization in coordinate measuring machine (CMM) operations. To evaluate whether the MBD design can save time compared to the current design method, two time studies are performed to compare the time spent in both of the design approaches.

2 Theoretical background of mechanical design and manufacturing

The following sections will introduce the background of mechanical design and how the design methods have been developing in recent history. After the mechanical design section, the most typical manufacturing methods of metal industry, computer integrated manufacturing and quality control are introduced.

2.1 Mechanical design in general

Initially, the need for transferring technical information came from the complex structures that people built. The traditional and most commonly applied manner to do that is to use technical drawings; they are a universal way to describe what to do and how to do it. (Quintana et al. 2010) From the designer's point of view, the techniques have been evolving remarkably from the drawing tables to the 3D-CAD. However, the final result of the design is basically the same as it was 50 years ago: a 2D-drawing.

The 2D-drawings are utilized in many different locations in the part supply chain. It is typical, that the design and manufacturing take place in different locations. Hence, the technical drawing must present the manufacturing information in a universal manner. Manufacturing information contains such technical information about the part that is not possible to be presented by a photograph or a written description. That technical information includes for example geometry, dimensions, material information and tolerances. (Hasari & Salonen, 2006)

2.2 Mechanical design details

The manufacturing of an exactly precise part is impossible meaning that the designer needs to make the decision on how precise the part needs to be in order to fulfill the requirements. In history, manufacturing methods were unsophisticated and parts were manufactured with wide tolerances. Hence, the assembly process usually required some modifications or "fitting", which usually required high skills from the assembly personnel. (Black, 2002, pp. 30-31)

As technology has evolved, the manufacturing of parts with tighter tolerances has become possible, which has offered certain benefits. First of all, the assembly process can be performed with less skilled personnel and reduced time, which leads to reduced costs and easier quality control. Secondly, repairing of a product becomes easier due to the possibility of replacing the faulty part with a similar part that is within the same tolerances. The third benefit is that the parts can be manufactured in large quantities, with less skills demanded of the machine operator. (Black, 2002, pp. 30-31)

Manufacturing with tight tolerances offers certain benefits. However, the use of tight tolerances increases the costs so significantly that the use of them needs to be closely considered. To be able to design cost-effective products a designer must be aware of how overly strict tolerances may increase the costs. (Black, 2002, pp. 30-31) Typical tolerances that a mechanical designer needs to apply to a 2D-drawing are: dimensional tolerance, geometrical tolerance and surface quality.

2.3 Evolution of mechanical design

Before the common use of computers, mechanical design was made manually to a paper. Few decades back, the first 2D computer aided drafting software were introduced and the drafting platform changed from a drafting board to a computer. Currently, a typical way is to do the design in 3D. Next chapters provide a background of each method and present the benefits and challenges encountered in each transition to a new method.

2.3.1 2D-drawing & 2D-CAD

Before the utilization of CAD-systems, the way to produce technical drawings was to do it manually without the help of computers. In order to be able to produce drawings that were readable and precise, the role of appropriate drawing tools was essential. Basic tools for manual technical drafting were pencil, pair of compasses, ruler and eraser. For professional use, different drafting apparatuses were utilized as well. (Pere, 2004, pp. 2:1-2:10)

The next step ahead from manual drafting was 2D-CAD. In the beginning, abbreviation "CAD" meant Computer Aided Drafting instead of the current meaning Computer Aided Design. Basically the implementation of 2D-CAD meant that the same work was done as before, but computer was used as a drafting platform. Even with that kind of arrangement, significant benefits could be achieved compared to the manual drafting. The most significant of the benefits were speed, quality and the possibility to integrate design to other computerized systems. (Pere, 2004, pp. 9:1-9:6) More detailed list of the benefits according to Pere (2004) is presented below:

- Increased automation. By utilizing 2D-CAD software the drafting could be partly automated. (E.g. section lines can be automatically added.)
- Changing and copying. Changing the features and repeating them was one of the benefits that was included in all of the CAD-software.
- Increased quality. Computer aided drafts have good appearance and they are exact.
- Reusability. Old drafts can be used as templates for new ones.
- Electronic archive. Electronic archive requires less physical space and is easier to control. Drawings do not wear out and can be printed out anytime needed.
- Standardization and modulation. CAD offers its benefits best when using as much standard items which supports the efforts towards rationalization and cost saving.

Even though 2D-CAD offered significant benefits compared to the traditional drafting, there were certain issues that needed to be taken into account. The list of the issues according to Pere (2004) is presented below:

- Hardware and software costs. Computers and software required to CAD-usage required money.
- The need for maintenance. To keep the software updated and diagnose and repair the problems, a CAD support person is required, which was a significant investment especially for the smaller companies.
- The need for education. To utilize the CAD-benefits properly, personnel needed to be educated. Further education might also be needed when updating the software.
- Transition costs. Learning the usage of the software required time and during the transition phase productivity of design might have decreased. (Pere, 2004, pp. 9:1-9:6)

2.3.2 3D-CAD

The introduction of 2D-CAD offered several benefits to the process and gradually replaced the drawing boards. The workflow of 2D-CAD went exactly as it went with manual drawings. Designer made each view separately and after the drafting, each view was annotated. Despite the large benefits it offered compared to manual drafting, there were still many issues that were not solved with the 2D-CAD software. (Popkov, 2014)

A 2D-drawing, whether it is made manually or by computer, can only describe the part by a limited number of projections and sections. This limitation can lead to misunderstandings especially when the complexity of parts is increasing. Another limitation is that a 2D-drawing is non-associative, which means that each change performed at any view of the part, needs to be manually repeated at the other views. It requires a lot of work and is a possible source of errors, because all the updates may not have been performed correctly. The third limitation relates to assembly drawings which are not linked to detailed part drawings. Hence, it is impossible to see while making the assembly drawing, whether the parts are compatible. The only possibility is to trust that the part drawings are dimensioned correctly. (Popkov, 2014)

With the help of 3D-CAD, the designer is able to use the model to visualize the product. Hence, the analyzing and documentation of the product requires less time and decreases the costs. By implementing 3D-CAD, the quality of the design is usually increased. The improvements originate from several sources including decreased design errors and the possibility to do finite element analysis. 3D-CAD implementation also provides an improved way to create 2D-drawings based on the model, making the drawings automatically correspond to the geometry of the model. Other quality improvements are achieved by offering more standardized drawings, better documentation, and fewer drawing errors. In addition to the previous, many pieces of the information provided during the design process can also be utilized in applications like CAM and NC-machining. (Lalit et al. 2008)

If the 3D-CAD is utilized efficiently, several benefits compared to 2D-CAD can be achieved. A list of the benefits according to Popkov (2014) is presented below:

- Automatic drawing creation from the model
- Automatic update of changes
- Automatic BOM creation
- 3D-visualization
- Reuse library for parts
- Parametric & template models
- Automatic mass calculation
- Interference & collision check
- Automatic flat pattern creation of sheet metal parts

Many of the benefits above affect the work of a designer making it more efficient and reducing the possibility of errors. From the manufacturing point of view, a benefit can be seen in the 3D-visualization, which can help in understanding the geometry. Another benefit is the improved quality of the drawings, which occurs because there is no possibility to do incorrect drawings as a result of the automatic drawing creation.

2.4 Design intent

Design intent is a term commonly used when discussing 3D-modeling and especially parametric modeling. There are several ways how to define design intent, but according to the most typical description design intent means how the model is built and how it behaves when a certain design parameter is changed. In more details, design intent contains all the relationships between the features and dimensions meaning for example sketch relations (parallel, horizontal, etc.) and feature references (e.g. hole referenced until next face). Usually the model history tree offers certain information about the design intent, but typically it is necessary to see details such as sketch constraints to capture the design intent completely. (Camba, et al., 2014)

2.5 Mechanical design process

A basic mechanical design process can be divided into two main phases: identifying the problem and solving the problem. The identification process can be further divided into three phases that are identifying the need, defining the problem and stating the limitations. The problem solving stage contains five different sub stages that are generating the ideas, evaluating the ideas, preliminary design, detail design and implementation. (Black, 2002, p. 91) Figure 4 shows the mechanical design process flow.

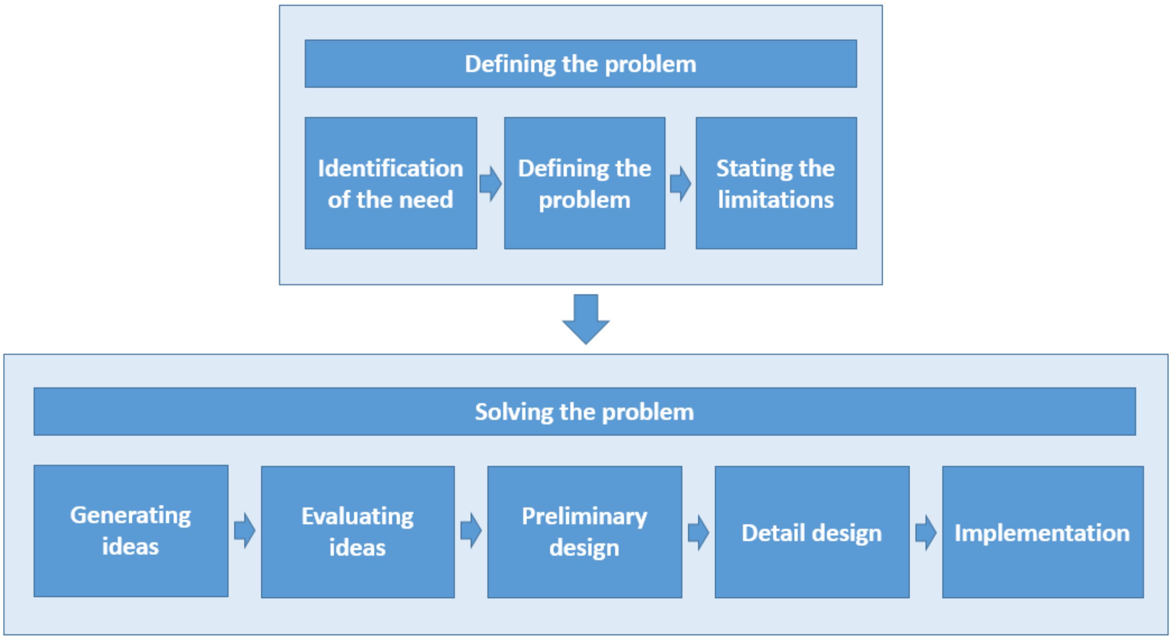


Figure 4 Mechanical design process flow (Black, 2002, p. 91)

In the beginning of the problem definition stage, the need for a certain product is defined. The purpose is to evaluate whether a true need for the product exists. The next steps in the process are defining the problem by quantifying the need and stating the limiting parameters. Basically these define the requirements of a product. (Black, 2002, p. 91)

The problem solving phase starts after the problem definition is completed. The problem solving begins with an idea generation in which the target is to propose several possible solutions to the problem. As a result of the idea generation, the best one of them will be selected as a solution. The next step is to begin the actual design phase containing preliminary and detailed design. After the detail design phase is complete the product will continue to manufacturing. (Black, 2002, p. 91) This thesis concentrates on the three last phases of the problem solving, because those are the phases where the 3D-design tools are present.

2.6 Typical manufacturing methods used in metal industry

To design a part which is functional and cost-effective, the knowledge about manufacturing methods, machines and tools is essential. The more designer is familiar with manufacturing methods, the more it is guaranteed that the design and dimensioning is suitable for a certain manufacturing method. (Pere, 2004, p. 7:45) The next chapters will provide a quick overview of some of the typical manufacturing methods used in metal industry and a slight review on how the methods are utilized among the Konecranes products.

2.6.1 Machining

The basic idea of machining is to remove material from raw material, which can be for example a casting or a forging. Even though castings and forgings are made to replicate the final product, usually they need to be machined to fulfill the technical or economical requirements. Machining directly from stock materials (e.g. tubes, plates and bars), is also a typical way of manufacturing. Even though the amount of waste material is usually higher, that approach provides certain benefits. First of all, the stock material is usually immediately available with short delivery times and will minimize the inventory costs and be cheaper in price per kilogram. In addition, stock material will more likely be usable for producing other parts in case the original product will not be manufactured anymore. (Black, 2002, pp. 193-197)

Machining is very widely utilized manufacturing method among Konecranes products. Typical parts that require machining include gearboxes, shafts, gear wheels and rope drums. In addition to the fact that they are common, those parts are also considered to be critical and are mostly manufactured by Konecranes factories rather than subcontractors.

2.6.2 Casting

As a manufacturing method, casting is appropriate for a very wide range of parts. The parts can vary from very small mass produced die castings to large structures that will be manufactured only once. The main benefit of casting is that it provides a half-finished part that is closer to the finished part thus reducing the cost of machining. In case the requirements of the finished part are loose or the casting method is accurate enough, the need of machining can be avoided. There are few characteristics of a part that will make it reasonable to be casted. A typical example is that the shape of the part is far from the stock materials. Another possibility is that the part is made of expensive material and minimizing the material waste is an important issue. (Black, 2002, pp. 187-190)

Casting is also utilized among Konecranes products. It is often used in case the volumes are large enough and the geometry is such that it is not easily or affordably achievable from the stock materials. Typical products that are manufactured by casting are gearboxes, bearing housings and chain hoist frames. Most of the castings require machining in the finishing phase to achieve adequate dimensional accuracy in functional features.

2.6.3 Forging

Similarly to castings, the basic idea of forging is to provide a half-finished part that is closer to the finished part than the stock material. Forging is generally suitable for more simple parts compared to casting. As an example car engine block is too complex to be forged, but a crankshaft can be either casted or forged. In certain cases, forging will provide benefits compared to casting. Unlike castings, forgings do not have internal defects that can exist in casted parts. In addition, forged parts have modified grain structure resulting from hot working, which will produce beneficial strength characteristics for certain types of parts. (Black, 2002, pp. 190-192)

Forging is a quite rarely utilized manufacturing method in Konecranes products. A product in which forging is utilized, is the hook. The hook forgings are typically standardized and the forgings are manufactured by subcontractors. Forgings, similarly to castings, usually require machining at the finishing phase.

2.6.4 Sheet metal manufacturing

The popularity of the utilization of sheet metal manufacturing originated from increased prices of materials. This issue set pressure to develop alternatives for machining, casting and forging. In addition, numerical control of bending and cutting machines made the usage of sheet metals efficient and flexible, and the increased quality of sheet metal materials has made the manufacturing method more competitive. (Ihalainen, et al., 1995, p. 232) Sheet metals can be divided into classes depending on their thickness. Konecranes classification is presented below:

- Thin plates < 3 mm
- Thick plates > 3 mm

There are several different sheet metal manufacturing methods including cutting, bending, molding and jointing. The cutting of sheet metals can be divided into thermal cutting (e.g. laser cutting) and mechanical cutting (e.g. perforation). A typical way of jointing sheet metals is welding, which is discussed in more details in the next chapter. (Matilainen, et al., 2011, p. 4).

Sheet metal manufacturing is very commonly used manufacturing method among Konecranes products. Thin and medium plates are generally utilized in various fixing parts while thick plates are typically utilized in load carrying frame structures. The sheet metal parts are typically manufactured by subcontractors and they rarely require machining.

2.6.5 Welding

The idea of welding is to join parts with or without additive material by melting the contact surfaces, utilizing intensive deformation or utilizing the diffusion (Ihalainen, et al., 1995, p. 281). As a result of welding, original parts form a united structure that has as good strength characteristics as the parent material (Black, 2002, p. 203). Welding is typically used in assemblies in which metal plates are assembled together. It is commonly used in large frame structures of thick metal plates.

2.6.6 Additive manufacturing

Additive manufacturing (AM), which is also known as 3D-printing, has gained popularity among media and researchers in recent years. The idea of additive manufacturing is to create

superposed cross-sectional layers of the part. Additive manufacturing process requires a 3D-model, which is sliced into thousands of layers (depending on the resolution) by preparation software and then printed layer by layer with appropriate material. (Gao, et al., 2015)
The term "additive manufacturing" was originally chosen by ASTM F42 committee. The term is supposed to differentiate the method from traditional machining in which the material is removed from the billet. Additive manufacturing has both benefits and drawbacks compared to the traditional manufacturing methods. Additive manufacturing offers the possibility to create complex shapes without the need of a mold, but it is relatively slow and expensive method. (Gao, et al., 2015)

Additive manufacturing is currently mainly used for making plastic prototypes. Individual plastic parts or small series can also be manufactured by the method. In addition, a study about utilization of 3D-printing of metals within spare parts business, is proceeding.

2.7 Assembly

To produce complete products, there is basically always need to do assemblies. The assemblies can contain parts manufactured inside the company, subcontracted parts or standard parts. The assembly work itself can also be performed either in-house or at a subcontractor. (Pere, 2004, pp. 16:1-16:2) In addition, the work as well can be performed by using various methods. Typical ways are manual assembly, automated assembly and robot assembly. (Boothroyd, et al., 2011)

To make the assembly work proceed smoothly, an assembly drawing which contains all the parts included, is necessary. Assembly drawings, and the assemblies itself, can be divided into three categories: main assembly drawing, assembly drawing and subassembly drawing. A drawing of a car, which contains all the parts included in it, is an example of a main assembly drawing. Respectively, a drawing of an engine of a car is an example of a basic assembly drawing, whereas a drawing of a single wheel is a subassembly drawing. (Pere, 2004, pp. 16:1-16:2)

The assembly work is typically conducted by bolt joints in which the work is mainly manual. Konecranes assemblies are typically quite large including large amount of parts. Hence, it is natural that there are several assembly levels meaning that a hoist assembly contains several levels of subassemblies.

2.8 Computer integrated manufacturing

Computer integrated manufacturing (CIM) began to develop after the second world war when Massachusetts Institute of Technology (MIT) began to develop CAD- and CAM-systems. The development need arose from the need to meet the design and manufacturing requirements of the aerospace industry. The manufacturing and design systems available in the late 40’s and early 50’s were not sophisticated enough to meet the requirements of the aircrafts and satellite launch vehicles of the time. (Radhakrishnan, et al., 2008, pp. 4-5)

The first part manufactured by a numerically controlled machine, was a supporting arch of an airplane wing. In the beginning of the NC-era, the machines operated without a memory or a processor and utilized punched tape in order to achieve the control parameters. The first machines that utilized computerized control were introduced in the 1970’s. Those machines were initially called CNC-machines (computerized numerical control) in order to separate them from the older machines. Currently, punched tape control is basically history and there is no longer need for the separation. A machine that utilizes numerical control is called an NC-machine. (Pikkarainen & Mustonen, 2010, pp. 13-20) Figure 5 shows the evolution of the numerical control in the first decades.

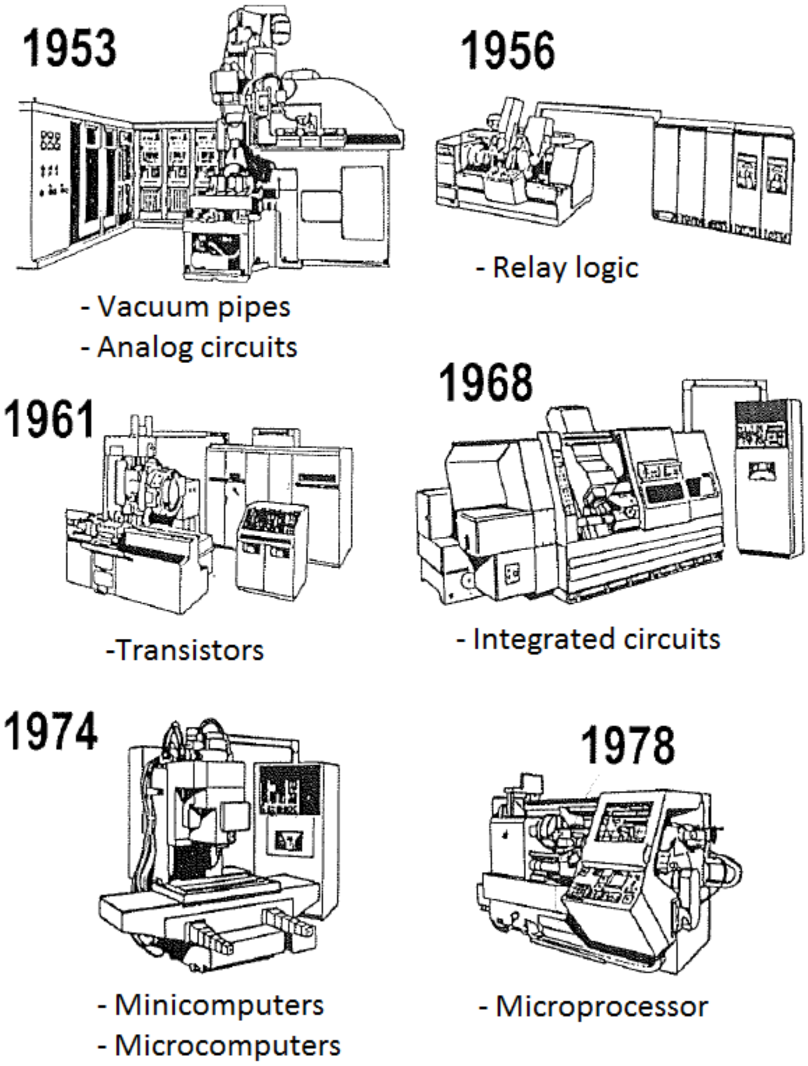


Figure 5 Evolution of the NC-machines in the first decades (Pikkarainen & Mustonen, 2010, p. 21)

2.9 NC-programming methods

To enable a good manufacturability of the part, the mechanical design needs to be complete. As stated in the previous chapters, mechanical design includes geometry information, tolerances and other manufacturing information about the part. That information should be transferred from the design to the NC-machine by an NC-program, which usually contains ASCII-text. The requirements of a proper NC-program are that it should be easily understood and editable by the programmer and by other possible users. In addition, it should be as efficient as possible to save machine time. It has been investigated that a decent NC-program can save up to 30 % of the machine time. Currently, there are three ways of creating an NC-program: manual programming, interactive workshop programming and computer aided programming. (Pikkarainen & Mustonen, 2010, pp. 102-106)

2.9.1 Manual NC-programming

Manual NC-programming is a method in which the program is generated manually based on a 2D-drawing of a part. The cutting paths are generated by calculating the coordinates of the tool location in certain time. When manual programming is used, the programmer should be familiar with the machine to be able to utilize it as well as possible. A skilled programmer can often make more efficient programs manually than by utilizing computer aided programming. Hence, manual programming is a usable method when the volumes are large and the parts are relatively simple. Usually the programming of lathes and bores is simple enough to enable manual programming. However, the programming becomes more complicated when utilizing machining centers or 5-axial machining. In addition, manual programming enables mistakes, which can be time consuming to debug or in worst case cause machine damages. (Pikkarainen & Mustonen, 2010, pp. 106-108)

2.9.2 Interactive workshop programming

Interactive workshop programming is performed straight to the machine by utilizing programming capabilities and interactive graphics of the machine. The control systems may contain macros, which can be utilized in order to describe the geometry of the part and the billet. In addition, they can help to choose the tools and their movements and by simulating the result. The workshop programming is usually used only for turning, because the geometry is rather easy, and it can be described in 2D on the display of a machine. The workshop programming is usually performed by the machine user, and it is a reasonable way when machining simple and small volume parts. (Pikkarainen & Mustonen, 2010, pp. 108-109)

2.9.3 Computer aided programming

Computer aided NC-programming utilizes the CAD-model of the part and produces the machining paths automatically based on it. The computer aided programming can be performed using either an integrated or a distributed system. An integrated system means that both the CAD-model and the NC-program are built using the same program, and the CAD-model contains the machining program. A distributed system uses separated CAD- and CAM-software and the model must be transferred from a CAD-system to a CAM-system prior to the creation of the NC-program. (Pikkarainen & Mustonen, 2010, p. 110)

The first step in creating the NC-program from a 3D-model is to import the CAD-model to the CAM-software. The CAM-software can be utilized in order to select the correct tool parameters, generate the tool paths and insert the process parameters. On the basis of these, the software then generates a CLData (Cutter Location Data), which must be translated by a post-processor to be usable by the machine. A post-processor is usually an independent software that is not part of the CAD- or CAM-software. (Majerik & Jambor, 2014) Figure 6 shows the process of creating an NC-program with a CAM-software.

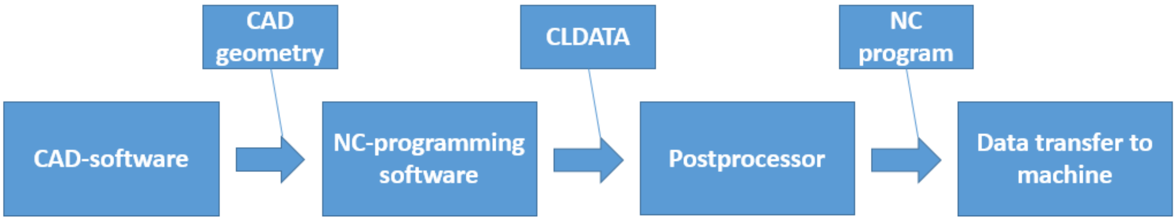


Figure 6 Computer aided NC-program creation process (Pikkarainen & Mustonen, 2010, p. 201)

Generally, the automated programming should be used when there is a CAD-model of the part, the parts are complicated, there are many different parts to be programmed and the machines are complicate. Manually made NC-programs are usually more efficient than automatically made, because an automatic program makes compromises in certain situations. However, automatically made programs are more faultless and quickly available. In addition, a skilled programmer can optimize the automatically made programs by making manual corrections to it. Those corrections may be relevant if the program is utilized for large volume products. (Pikkarainen & Mustonen, 2010, p. 110)

2.10 Quality control

The basis of the manufacturing is to make changes to raw material and thus achieve a required part. However, decent parts can only be manufactured if the process is supervised and controlled constantly. The methods, which ensure that the part corresponds to the design are called quality control. (Black, 2002, p. 174)

A major benefit of an NC-machine is that it is capable of manufacturing multiple parts with equal dimensions. Basically, the only issue that can cause variation in dimensions is the wearing of the tool. The quality control can be performed by using basic tools such as slide gages, micrometers and thickness gages. However, there are also more sophisticated measuring machines that can perform the measurements automatically based on a 3D-model of a part. In addition to the other phases of the production, quality control is a source of cost. Hence, only the first parts of an NC-machined series are typically inspected carefully until a steady state is achieved. After that, only occasional checks are generally performed to ensure that the production goes as planned. (Pikkarainen & Mustonen, 2010, p. 216)

The parts are inspected to assure that they fulfill the requirements set by the designer. In a mechanical part those issues include dimensions, surface quality and geometric tolerances. Common stages, in which those inspections are performed are an inspection of the incoming materials (e.g. raw material or subcontracted parts), an inspection between the manufacturing stages, an inspection of a complete part and an inspection of a complete assembly. (Radhakrishnan, et al., 2008, p. 508)

As mentioned earlier, there are several ways to perform the quality checks using relevant machines. Certain measuring machines can generate a 3D-representation of the actual part, which is basically a reverse-engineered model of it. Computer aided measuring machines can be divided into two classes: contact and non-contact machines. A typical example of a contact machine is the coordinate measuring machine (CMM) and a typical example of a non-contact machine is the laser scanner. The CMM utilizes sensing probe in order to achieve the geometry while the laser scanners and other non-contact machines usually measure the radiation reflection. (Radhakrishnan, et al., 2008, pp. 510-515)

2.11 Conclusions

As a conclusion, both mechanical design and manufacturing are developing towards more digitalized environment. From the mechanical design point of view, the CAD-tools and the 3D-models are continuously developing and new possibilities will be available constantly. In addition to the design, the manufacturing methods, machines and software are also developing towards more automated manufacturing. Because each manufacturing department needs the information generated in mechanical design, it is beneficial if the information can be conveyed as automatically as possible. Because the model created in the design department is utilized in several functions including manufacturing and quality control, it is important that all the different software utilized within the operations are working seamlessly together.

3 Model-based Definition (MBD) – Literature review

Traditionally, the geometry and manufacturing information of the part have been transferred from a design department to a manufacturing department by using 2D-drawings. In recent years, a new method, which includes all the product and manufacturing information (PMI) straight to the 3D-model and replaces the 2D-drawing, has been introduced. There are two commonly used terms when discussing about a drawing-less environment and 3D-annotations: model-based definition (MBD) and model based enterprise (MBE). Model-based definition means an approach to design, in which all the design information is defined at the 3D-model. Model-based enterprise means that the same 3D-model is utilized throughout the whole value chain. (Manninen, 2015)

The following chapters provide a literature review of model-based definition. Chapters provide information from several sections including MBD background, MBD characteristics, MBD related standards and MBD related file formats.

3.1 Background

As mentioned earlier, 2D-drawings are still generally utilized as a main medium of transferring the manufacturing information. From the technical point of view, it is currently possible to include the annotations directly to the 3D-model of the part, allowing to phase out the use of a 2D-drawing. (Quintana, et al., 2010) Information required to fully define a part include geometric dimensioning and tolerancing (GD&T), surface quality specifications, material specifications etc. That information is called product and manufacturing information (PMI). (Frechette, et al., 2013)

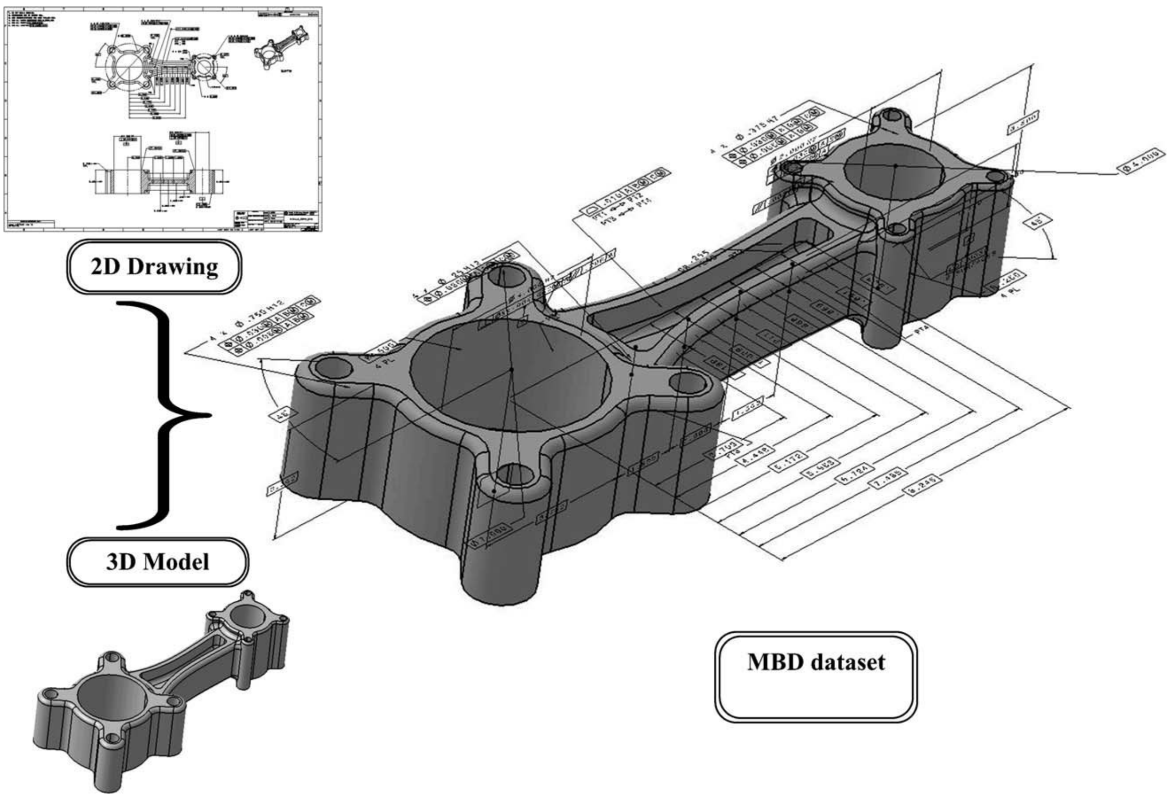


Figure 7 Annotated 3D model (Quintana, et al., 2010)

Figure 7 defines the basic idea of MBD. Instead of the traditional two datasets, a 3D-model that defines the geometry and a 2D-drawing that defines the manufacturing information, all the necessary information is included in an MBD-dataset. ASME Y14.41 was the first standard that provided standardized rules for displaying and orientating the annotations in a 3D-model. (Quintana, et al., 2010)

3.2 MBD characteristics

There are several possible benefits resulting from MBD. As stated earlier, the current design practice is usually to create a 3D-model and later a 2D-drawing based on the model. In many cases, only the 2D-drawing is used in the further operations. In the MBD approach, only an annotated 3D-model will exist, which eliminates the possible conflicts between the model and the drawing. In addition, it eliminates the need to store and control two separate files, thus saving time and database space. The benefit of an annotated 3D-model is higher when the complexity of the part increases. Understanding a complex geometry based on a 2D-drawing requires high technical expertise, while understanding a geometry in a 3D-form requires basically no technical knowledge. The MBD dataset also contains implicit information meaning that the user can for example take measurements of the features that are not dimensioned, or make an additional section view. (Quintana, et al., 2010)

In addition, the 3D-annotations can be shown selectively so that the user only sees the annotation that is desired. The annotations are also associative, meaning that they update automatically when the geometry changes. The MBD tools also prevent the misuse of the geometric tolerances, meaning that for example perpendicular tolerance cannot be added to two surfaces that are not perpendicular. An additional benefit of the 3D-annotations is that they can be utilized in tolerance stack up analyses. In addition, an annotated 3D-model can be used in an automatic creation of a CMM-program thus saving time and preventing errors in the inspection process. By utilizing MBD, the main dimensions can be immediately added to the 3D-model instead of waiting for the drawing to be produced. Thus the design intent is constantly visible. (Quintana, et al., 2010) Because an annotated 3D-model eliminates the need to create a 2D-drawing, it has been estimated that in an MDB environment, 30 % of engineering time can be saved. (Rapinoja, 2015)

The transition to model-based definition requires changes throughout the whole supply chain. The design work must be performed in a disciplined way to achieve a model that is usable in different processes. To make the annotation process as simple as possible, the features of a 3D-model should be modeled as they will be manufactured in reality. For example a group of threaded holes should be modeled as a group of holes that have a thread as a parameter. Thus, the annotation work can be performed correctly and as automatically as possible. The annotated 3D-models require a different approach also from the personnel that handles the models, because the 2D-drawings no longer exist. However, annotated 3D-models are generally better and less error prone way to convey the design information, because the geometry can be more easily understood. Certain query actions can also be applied to the models, in which for example geometric tolerance relationships can be made clear. In addition, 2D-drawings can quite easily be generated based on an annotated 3D-models, whether certain action requires the drawings. (Rapinoja, 2015)

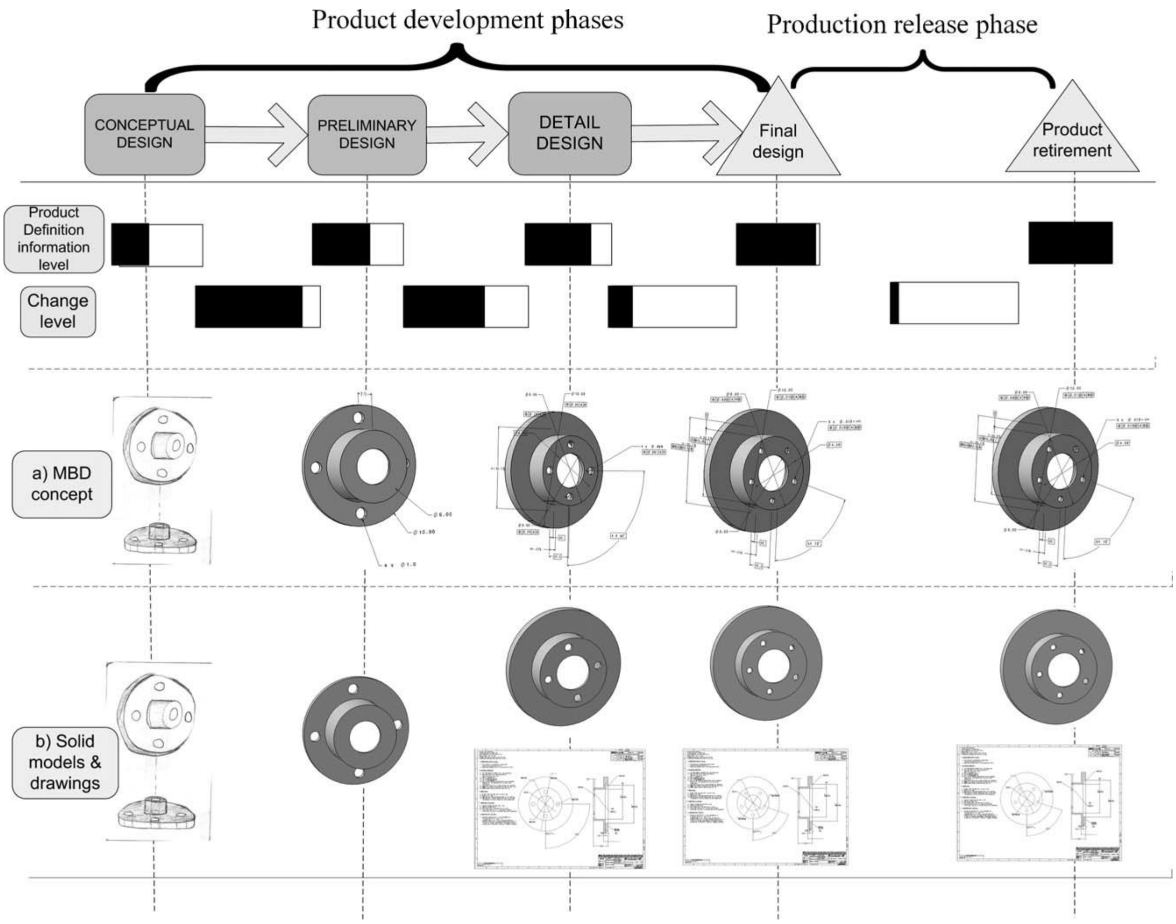


Figure 8 Comparison of product development cycles of MBD and traditional approach (Quintana, et al., 2010)

Figure 8 determines the differences between the product development cycle of the MBD concept and a concept that utilizes both 3D-model and a 2D-drawing. Basic difference of the approaches is that the MBD concept relies on a single model that carries all the product definition data, while the traditional approach has two different files for the same purpose. Basically, the processes proceed similarly until the detail design phase, in which the 2D-drawing is created in the traditional approach. Minor difference can also be detected at the preliminary design phase, in which certain preliminary dimensioning can be performed in the MBD approach. (Quintana, et al., 2010)

By looking at Figure 8 it can be said that the MBD approach simplifies the work of a designer at least in the cases where certain design changes need to be performed. When a feature of a 3D-model is changed or removed, the annotation changes can immediately be seen in the model instead of updating the drawing separately. In addition to the possible time savings, this also removes the possibility of errors that occur, if the model and the drawing are not updated correctly and do not correspond to each other.

3.3 Scale of usage and history of MBD

The history of MBD reaches to 1990's, when United States defense forces began the development of it. The latest military standard MIL-STD-31000A was introduced in 2013 and to follow the standard, the usage of MBD is required. Since the beginning of year 2014, MBD became the recommended document sharing practice for all the suppliers of U.S.

defense forces. In addition to U.S. defense forces, also other major actors such as Boeing, Ford and Toyota have been implementing the MBD practices to their design and manufacturing chains. Due to the matter that major companies are implementing the MBD practice to their suppliers, the practice is spreading from major companies to medium-sized and small companies. (Manninen, 2015)

Basically, many companies are already operating at least partly according to the MBD approach. Machining paths can be acquired from a 3D-model and the 2D-drawings operate only as a confirmation of the manufacturing information. In addition, the current standards, software and methods are compatible with the MBD environment. That supports small- and medium-sized companies, which do not have major resources, in order to develop their practices towards the MBD approach. (Manninen, 2015)

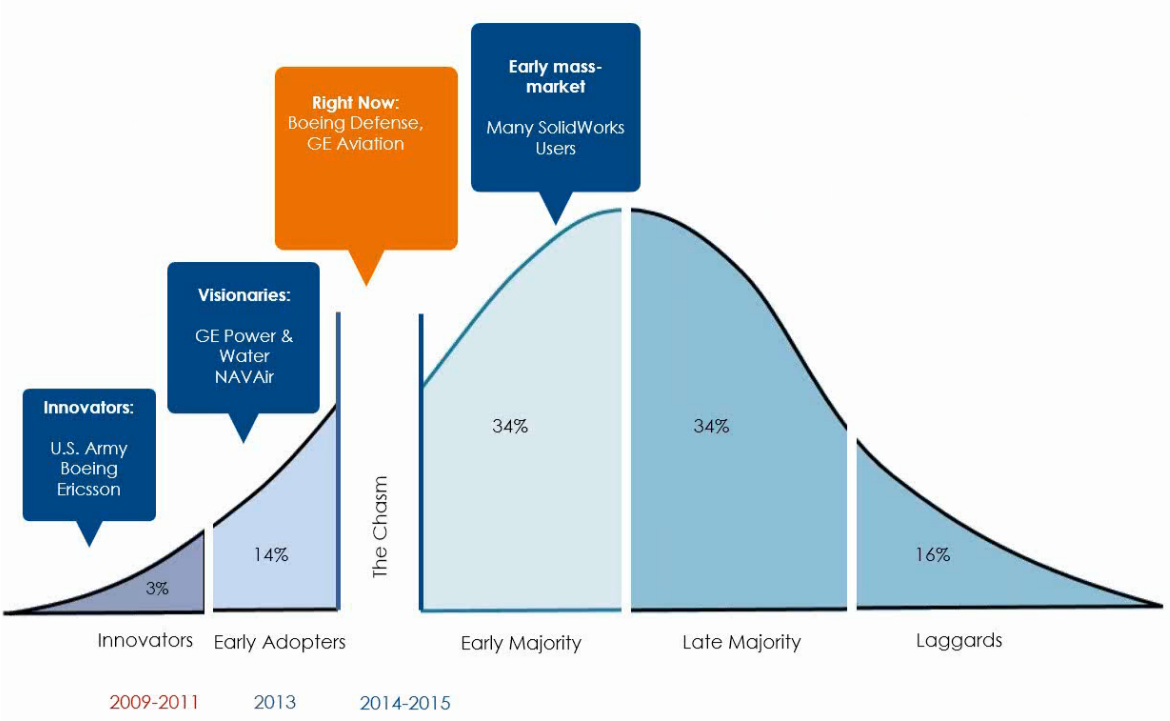


Figure 9 Evolution of MBD usage within CAD-users (Manninen, 2015)

Figure 9 shows the innovation curve, which presents the percentage of the users of a new method. Currently, the status is that the innovators are already fully using the system and the early adopters are starting to have their systems ready for the new practice. Development orientated early majority is currently beginning the implementation of the MBD practices. CAD-software SolidWorks executed a research in 2009 and 2012, for 700 users about their plans to introduce MBD in their functions in the next three years. The results were as follows:

- 2009: 1,5 %
- 2012: 16 %

Those percentages are quite closely in line with the innovation curve. Currently (2015), according to innovation curve, the percentage would be 17 %, compared to the 16 % achieved from the research. (Manninen, 2015)

3.4 MBD user experiences

The positive user experiences from the companies that have introduced MBD work as an important reference for the companies that are considering, whether to develop their practices towards the MBD-environment. However, the percentages of cost-savings and verbal feedback of MBD are generally so significant and positive that certain criticalness needs to be kept in mind. In addition, the companies that have invested to the technology, want to show that they have made a correct decision. The user experiences are also delivered through CAD-suppliers that have their own interests about the issue.

One of the companies that has utilized the MBD approach is Saab Aeronautics. Saab Aeronautics is a business area of the Swedish company Saab and it operates within the development of military and civil aviation technology. One of their products is the Gripen fighter. (SAAB, 2015) According to a PLM Group webinar presented by Manninen (2015), Lars Ydreskog from Saab Aeronautics has stated that they have utilized MBD in several product development projects. Product definition, mechanical design and the models of the manufacturing tools have been completely implemented with the 3D-tools. Benefits they have achieved are 25-35 % decrease in product development times and decreased number of documents. In addition, they said that the only way to compete with larger companies is to supply material in a 3D-form. (Manninen, 2015)

Another major organization that utilizes MBD is the United States defense forces. The office of the U.S. Secretary of Defense (OSD) has declared following experiences that recommend the usage of MBD. One third of design costs arise from the creation and upkeep of the 2D-drawings. However, 60 % of the drawings do not correspond to the 3D-models. U.S. Department of Defense has also stated the benefits that have been achieved by MBD. The first benefit is that the unproductive work in design has decreased by 30 %. In addition, defects and reparations have been decreasing by 20 %. The last benefit stated, is that the response time of subcontractors has been decreased by 50 %. (Manninen, 2015)

Two additional companies that have announced benefits resulting from MBD are Toyota and Boeing. Both of them have declared that MBD has reduced the time used in the manufacturing tool design and the manufacturing by 50 %. There has also been a significant reduction in the assembly lead times, which have decreased by 33 % at Toyota and by 30 % at Boeing. An additional benefit for both of them has also been the decrease in reparations. The percentage at Toyota has been 30 % and at Boeing 50 %. (Manninen, 2015)

3.5 MBD related design standards

There are several standards that are related to MBD. The first standard that establishes the requirements for an annotated 3D-model was an American standard ASME Y14.41. It was initially issued in 2003 and revised in 2012. The need for the standard was originated by the wish to utilize CAD-data within manufacturing and inspections. Especially, the complex structures utilized in aerodynamics and ergonomics, made the usage of a 2D-drawing difficult (Model Based Enterprise, 2014). International standard ISO 16792, which is based on the ASME-standard, was initially issued in 2006. The latest revision, that also has a Finnish version SFS-ISO 16792, was issued in January 2016. Another standard that is made based on the ASME Y14.41 is U.S. military standard MIL-STD-31000. It was originally issued in 2009 and revised in 2013.

Other standards that relate to MBD are for example SFS-EN-ISO 1101 and SFS-EN-ISO 5549. SFS-EN-ISO 1101 defines the basic principles and requirements of the geometric tolerancing for both 2D- and 3D-annotations. SFS-EN-ISO 5549 relates also to the geometric tolerances by defining the basic elements. It is currently under revision and the revised version will also support the annotated 3D-models (Rapinoja, 2015). This thesis gives a more detailed review about the standards SFS-ISO 16792 and MIL-STD-31000.

3.5.1 SFS-ISO 16792

SFS-ISO 16792 is made based on the ASME Y14.41 and the purpose of the standard is to provide a sufficient representation for an annotated 3D-model to correspond for the requirements of a conventional 2D-drawing. The standard takes into account that the 2D-drawings will be still used in the future and it supports two ways of the data representation: only 3D-model and a 3D-model with an associative drawing. (SFS-ISO 16792, 2010)

The Standard is very wide and it defines the requirements of an annotated 3D-model extensively. Some of the requirements are introduced in the following paragraphs. The complete data set that defines the product should include a 3D-model, annotations and related documents. According to the standard, the annotations can be placed either on the 3D-model or on the 2D-drawing. There are several requirements for the annotated 3D-model including that all the annotations must be attached to one or more planes that follow the model when it is rotated. Another example requirement is that the annotations must be readable regardless to the position of the model, meaning that the annotation reading direction must update after a rotation. An additional requirement for the 3D-annotations is that the display of the annotation must be possible to be enabled or disabled with three options as follows: all the dimensions visible (Figure 10), the dimensions of one type visible (Figure 11) or the selected dimensions visible (Figure 12). (SFS-ISO 16792, 2010)

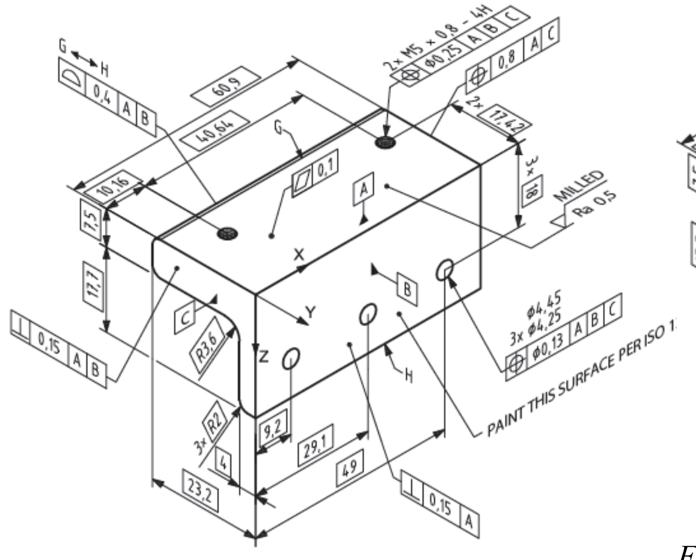


Figure 10 All the annotations visible (SFS-ISO 16792, 2010)

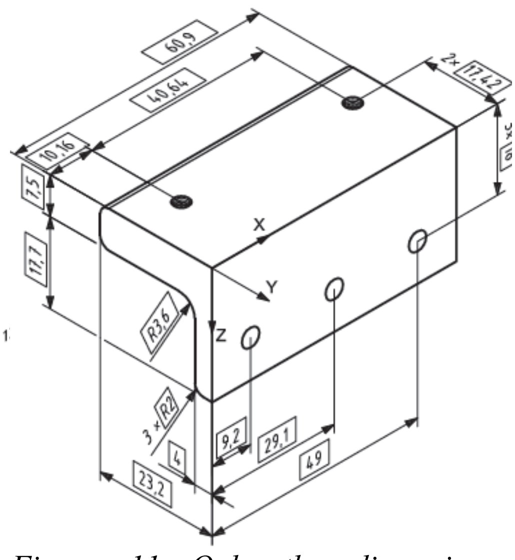


Figure 11 Only the dimension annotations visible (SFS-ISO 16792, 2010)

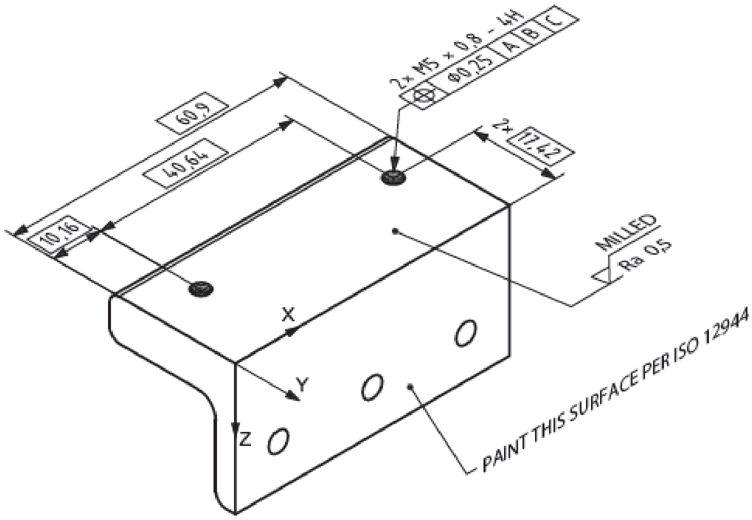


Figure 12 Selected annotations visible (SFS-ISO 16792, 2010)

An important feature that the standard defines is the possibility to apply queries to the model. Basically, a query means that the user selects an annotation and all the features associated to the annotation are highlighted in the model. There are several annotation types in which the queries should be applicable including geometric tolerances, surface qualities and hole patterns. In case an annotation of a geometrical tolerance is selected, the associated reference datum is highlighted from the model. By selecting a surface quality annotation from the model, the associated surface is highlighted. Figure 13 presents how the highlighting works when selecting the annotation of a hole pattern. (SFS-ISO 16792, 2010)

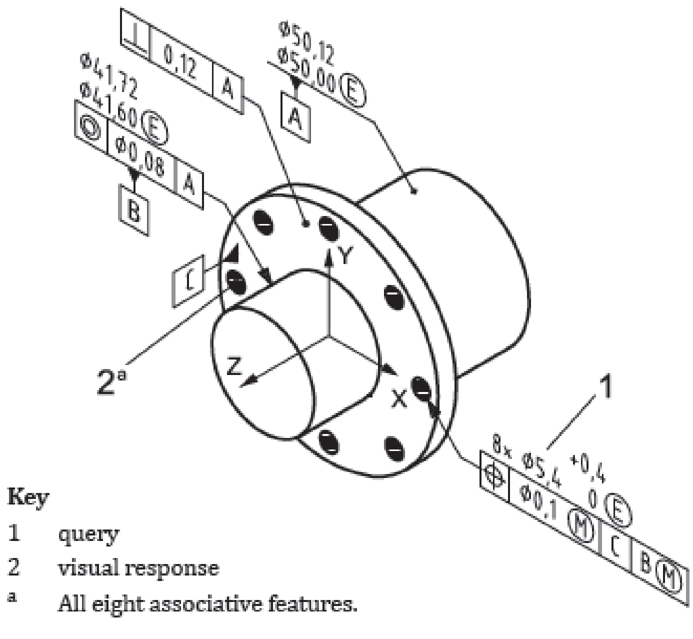


Figure 13 A query of an annotation and a visual response (SFS-ISO 16792, 2010)

3.5.2 MIL-STD-31000A

The MIL-STD-31000A was introduced in 2013 in order to contribute to the modernization of data acquiring practices of the U.S. Department of Defense (DoD). The main content that the standard provides is the definition of a technical data package (TDP) and how it should be utilized throughout the product lifecycle efficiently. The standard defines that the TDP is "A technical description of an item adequate for supporting an acquisition, production, engineering, and logistics support (e.g. Engineering Data for Provisioning, Training, and Technical Manuals)." The standard defines three different TDPs: a 2D-TDP, a 3D-TDP with the model and the associated drawings and a 3D-TDP with only the model, which contains all the required information. (MIL-STD-31000A, 2013) (Whittenburg & Whittenburg, 2013)

In addition to the TDP representation, the standard also defines three different levels of information required in different phases of the product development. The first level of the TDP is a conceptual level, which consists of a graphical representation of a concept that is necessary in order to evaluate and analyze the concept. A development level TDP is defined so that it includes enough information for prototyping or for a limited production. A production level TDP contains all the information needed in order to manufacture the part, without the contribution of the original designer. (MIL-STD-31000A, 2013) (Whittenburg & Whittenburg, 2013)

Appendix B of the standard introduces a modeling schema in order to assure that the models are reusable and easily understandable. Traditionally, the 3D-model has had an accompanying 2D-drawing, but with MBD all the product definition information can be included in the same 3D-model. Figure 14 shows the journey of how the documentation of the TDP has evolved. (MIL-STD-31000A, 2013) (Whittenburg & Whittenburg, 2013)

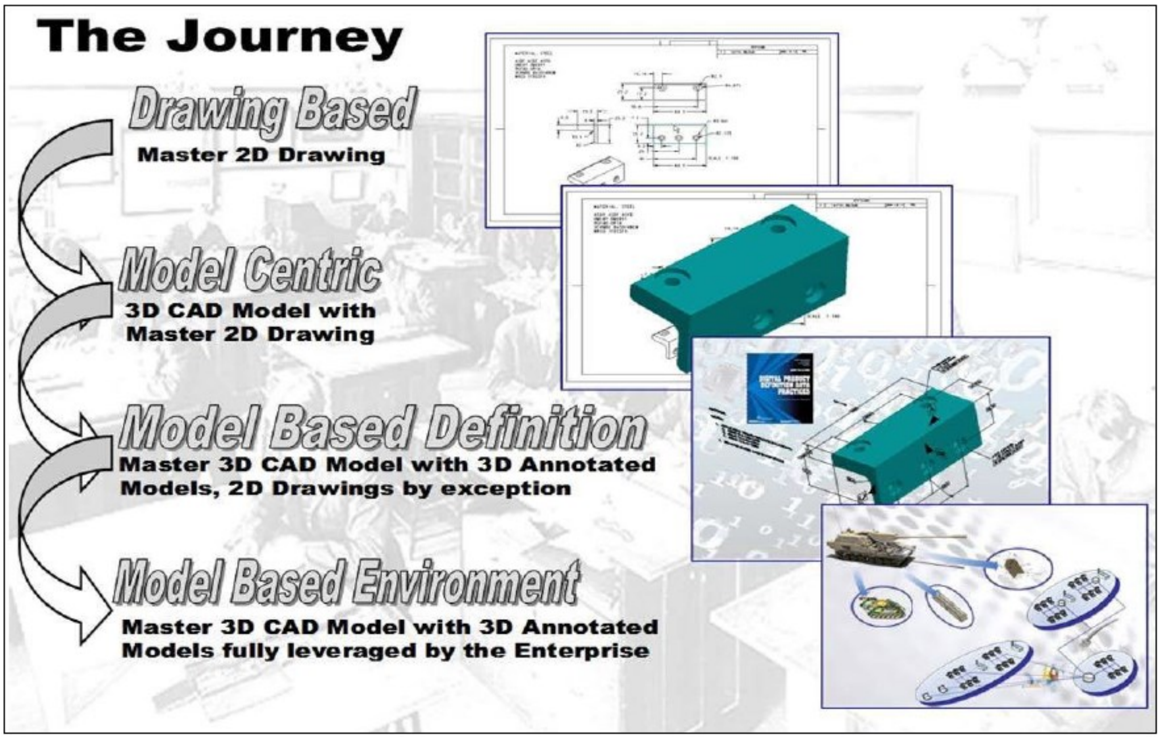


Figure 14 Evolution journey of a TDP (MIL-STD-31000A, 2013)

The appendix B also defines three different levels of annotation that define the amount of details in the model. A minimally annotated model can be utilized for laser cuts or prototypes, and it includes only the envelope dimensions, general tolerance, material, finishing requirements and the title block information. The standard defines that a partially annotated model could be used when the product has secondary operations and is used for primary operations. In addition to the annotations on the minimally annotated model, the partial annotation includes non-standard dimensions, site map and critical notes. The full annotation means that the model must include full dimensioning and auxiliary views. It should be utilized if the product is to be manufactured by an external supplier. Figures 15-17 present the different annotation levels. (MIL-STD-31000A, 2013) (Whittenburg & Whittenburg, 2013)

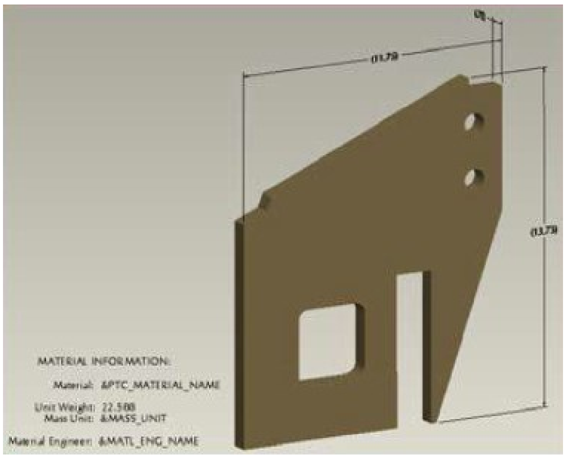


Figure 15 Minimal annotation (MIL-STD-31000A, 2013)

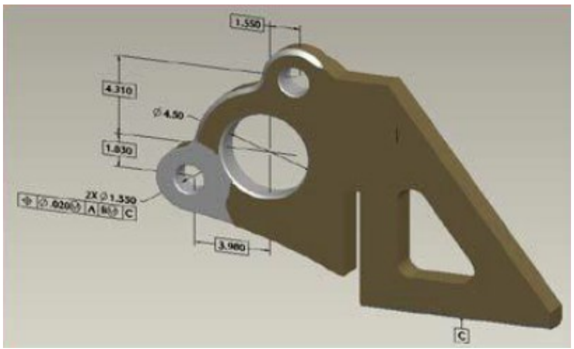


Figure 16 Partial annotation (MIL-STD-31000A, 2013)

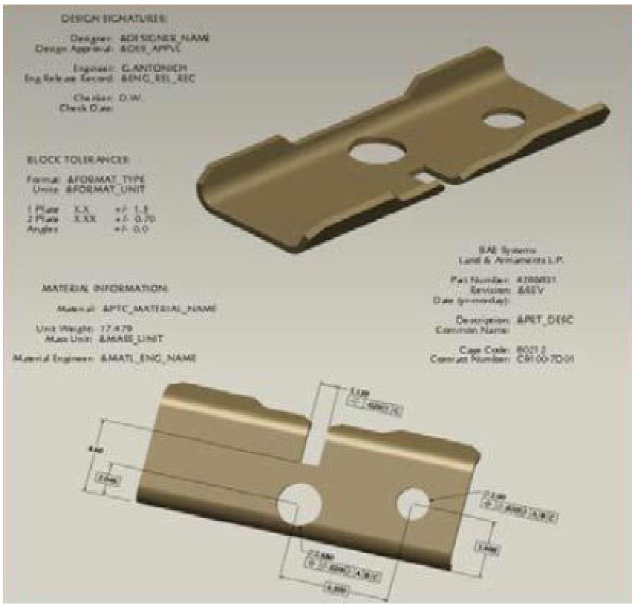


Figure 17 Full annotation (MIL-STD-31000A, 2013)

3.6 MBD-related file formats and data exchange

There are several different systems and software that manage the data in the design and manufacturing environment. Traditionally, each system has had its own format and the data has had to be applied separately to each system. This causes unnecessary work and is a potential source of errors. Many solutions for the issue have been proposed over the years of which the most successful have been the standards for the data exchange.

There are several standards defining how the product data can be exchanged between different software and how the 3D-models can be visualized. Successful formats include for example ISO 10303 (STEP), ISO 14306 (JT) and ISO 14739 (3D-PDF). Even though the exchange of the geometry data with neutral formats works quite well, the exchange of the manufacturing data has not been as successful due to certain issues that are encountered in the data exchange process. The first issue is that the manufacturing processes are typically highly customized and the data used in a certain process might be useless in any other manufacturing facility. Another issue is that the manufacturing process data is considered to be confidential, which decreases the willingness to share the data with external suppliers and customers. The third reason is that there are only few neutral formats for manufacturing data and native formats are too specified for certain machines in order to be utilized generally. (Lubell, et al., 2013)

If a standardized way to exchange the manufacturing process data could be achieved, it would offer large benefits. The basic lines of the manufacturing processes could be included in the model and the detailed process planning could be left to the manufacturer. This would decrease the amount of total work, because the basic process is typically almost similar and the designer could apply the basic information to the model. One of the standards offering information about the manufacturing processes is ISO 10303-238 (STEP-NC), which has been used to exchange the manufacturing programs. ISO 13399 is a standard for cutting tool data representation as well as data exchange, and ISO/DIS 10303-242 (STEP AP242) includes information about assembly tolerances, surface finish and manufacturing information. Those standards could be used together to help in automating the manufacturing processes and sharing the process data through the model-based enterprise. (Lubell, et al., 2013)

3.6.1 JT

JT is a 3D data format, which is developed by Siemens PLM Software. JT can be used for visualization, collaboration and 3D-data exchange. (Lubell, et al., 2013) JT was the first format for viewing and sharing of lightweight 3D product data, which was accepted as an ISO standard. (Siemens PLM Software, 2012) In addition to the product geometry, the JT-file can contain product manufacturing information, product structure and model metadata (attributes). A JT-file can be exported from the native CAD-software and information can be added to a JT-file from other applications such as PDMs. (Keating, 2012)

JT open program is a community of users, software vendors and other interested parties that share information of JT and influence in the development. The program has a goal to increase the usage of JT in visualization, collaboration, interoperability and data archiving. There is a free to download no-cost JT viewer available through the program. (Keating, 2012) The JT viewer is called JT2Go and according to Sicking (2014), the software has following functionality:

- Viewing of 2D- and 3D-files
- 3D-measuring and sectioning
- Displaying product manufacturing information
- Embedding JT to Microsoft Office documents
- Display of CAE results

Figure 18 presents the functionalities of the software JT2Go.

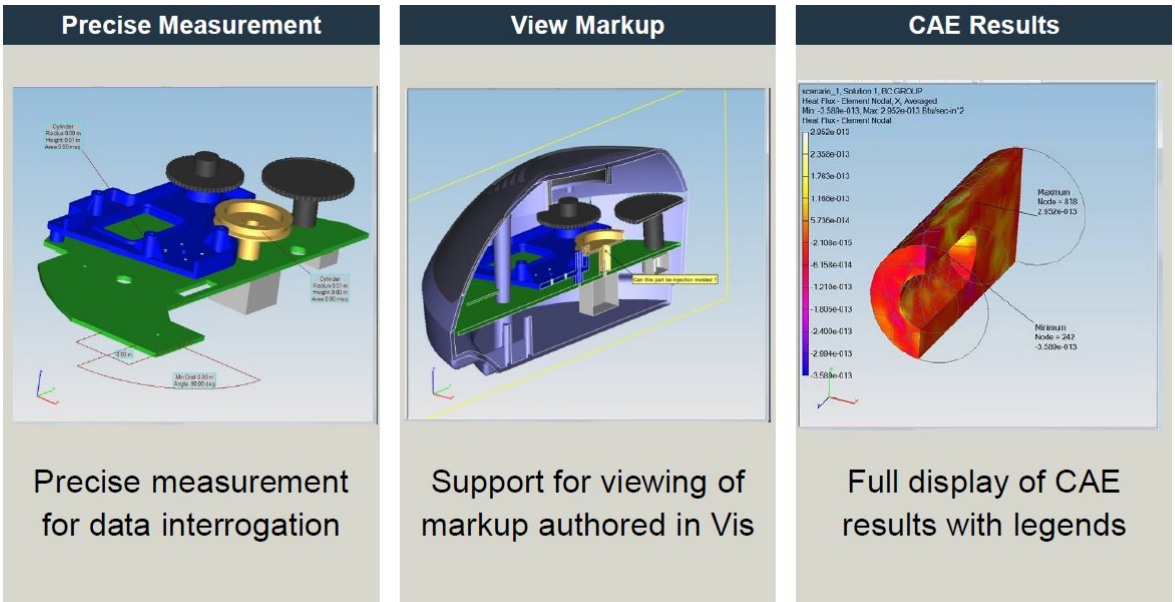


Figure 18 Features of JT2Go software (Sicking, 2014)

3.6.2 STEP

STEP is actually not a format, but an international standard (ISO 10303) that consists of several application protocols (AP) and is the largest standard of ISO. The standard is defined to be a "standard for the computer-interpretable representation and exchange of product manufacturing information". It was originally generated by the aerospace industry when there was a need to exchange data between the airframe manufacturers and their suppliers in an organized form. Later it has expanded to other industries as well, and includes additional domains beyond to geometry. (Opsahl, 2013) Some of the most important areas that the STEP covers according to Opsahl (2013), are listed below:

- Geometric information
- Product manufacturing information
- Product lifecycle information
- NC-manufacturing information
- Electrical systems information

STEP 203 & STEP 214

The two most widely implemented STEP standards are AP 203, which is maintained by company PDES, Inc. and AP 214, which is maintained by companies ProSTEP iViP and SASIG. The AP 203 is primarily supported by the aerospace and defense industries, while the AP 214 is primarily supported by the automotive industry. There are certain differences between the two standards, but many important issues are included in both of them. The issues included in the both standards are for example 3D PMI, 3D solid geometry, product management data and attributes. (AIA; ASD; PDS, Inc; ProSTEP iViP, 2009)

STEP 242

STEP AP 242 is called "Managed Model Based 3D engineering", and the basic idea of it is to merge the two most commonly used STEP standards AP 203 and AP 214. Previously the two standards have covered partly the same areas, which has created a need to optimize the resources at standard development and maintenance. An additional purpose of the AP 242 is to enable and standardize the smart manufacturing operations meaning basically machine readable models that contain the PMI. One of the use cases for AP 242 is the data exchange and interoperability between the different CAD-software, in which the previous STEP standards have been operating successfully. AP 242 contains all the features contained in AP 203 and AP 214 and certain additional features as well. It should also be capable of providing reusable data for CAM- and CMM-solutions, as the providers of the systems are currently working on enabling AP 242 PMI for automated processes. It has also been stated that STEP AP 242 would be stable for the long term archival for over 70 years. (Feeney & Hedberg, 2014)

STEP-NC

Purpose of the STEP-NC, ISO 10303-238 *Application interpreted model for computerized numerical controllers* (AP238) is to provide data that is interoperable between CAM-software and machine tool controllers of NC-machines. The STEP architecture supports the data transfer between the different APs. Therefore product geometry may be defined in e.g. format AP214 and manufacturing features in AP 224 (*Mechanical product definition for process planning using machining features*). This data can later be utilized by a CAM-software, which outputs geometry, features, process sequence and tool requirement data in the STEP-NC format. The geometry does not need to be regenerated, as it is possible to reuse the original model. The STEP-NC data is processed at run-time, which means that the specific machine operations, such as cutter paths, are performed by the machine controller. Hence, the STEP-NC is not machine specific and can be utilized on any machine that fulfills the tooling requirements. (Feeney & Frechette, 2002) Figure 19 shows the location of the STEP-NC in the part creation process.

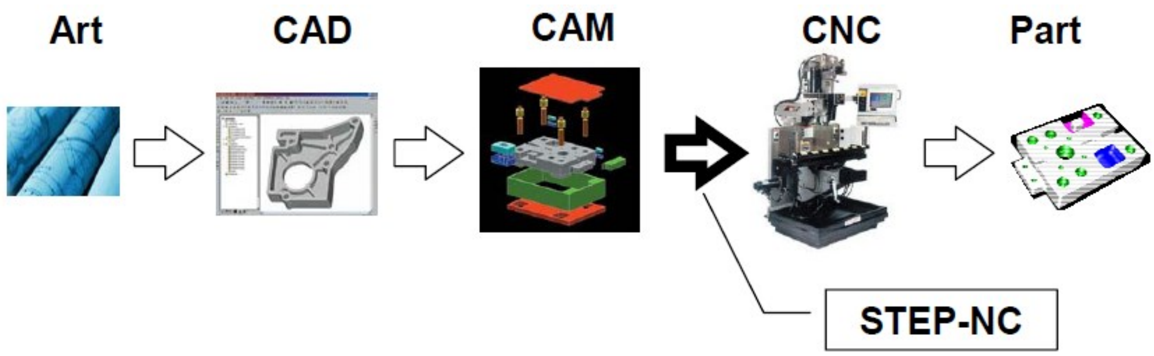


Figure 19 Location of a STEP-NC-file in the process (Feeney & Frechette, 2002)

3.6.3 IGES

The history of IGES reaches back to year 1979 and Air Force ICAM Industry days meeting. An issue that was encountered was that the data exchange between different CAD-software was not possible. After discussing about the issue, an organization that was named Interim Graphics Exchange Specification (IGES), was formed. The first draft, which contained geometry, graphical data and annotations, was presented in 1980 and in 1981 IGES became an ANSI standard Y14.26M. (Kemmerer, 1999)

An IGES file consists of information structures or entities that are used to represent and communicate the product data. Each entity is represented in an application independent format that is readable by most of the CAD- and CAM-software. The entities can be divided into geometric and non-geometric entities. The geometric entities represent the physical shape of the part including curves, points, surfaces, solids and relations. The non-geometry entities are used to provide attributes for the individual entities. The attributes include views, drawings, dimensions, texts, notations etc. (Bhandankar, et al., 2000)

IGES is a neutral format, which can be utilized in representation and exchange of the product data in an electronic form. However, it has certain problems that are restricting the usability. First of all, it does not have a formal data model, which may lead to ambiguities. Secondly the data exchange might be incomplete and data might be lost, because IGES lacks conformance requirements and software vendors have added different "flavors" to the files. The third problem is that IGES only supports drawings and 3D-models, excluding other life cycle data such as manufacturing information. The last problem is that IGES file format structure has 80 columns, which makes it difficult to understand and edit. (Bhandankar, et al., 2000)

3.6.4 3D-PDF

Portable data format (PDF), was specified by Adobe in 1993 and it remained as a proprietary format until 2008, when it was released as an open standard (ISO 32000-1). The PDF itself does not define a 3D-data standard, but it supports two types of 3D-representation. In 2005, PDF started to support Universal 3D (U3D) format which is an ECMA standard that was developed by an industry group called the 3D Industry Forum (3DIF), including companies such as Boeing and Intel. U3D supports only tessellated data (non-exact geometry constructed from e.g. tetrahedrons), product structure, animations and textures. It does not support exact or boundary representation (BREP) geometry or product manufacturing information. (Opsahl, 2013)

In 2006 Adobe acquired a company called Trade and Technologies France (TTF), which was developing a digital mockup application called 3Dreviewer. TTF owned significant pieces of the technology and intellectual property, including libraries, which allowed many data formats to be imported into a format called Product Representation Compact (PRC). The formats included major CAD-formats from Catia, ProE and NX. In addition to the tessellated data and the product structure provided by U3D, PRC also provides a support for exact geometry and PMI. U3D and PRC are both supported by Adobe Acrobat and are specified by ISO 32000 (PDF). (Opsahl, 2013) Figure 20 presents a 3D-PDF and its contents.

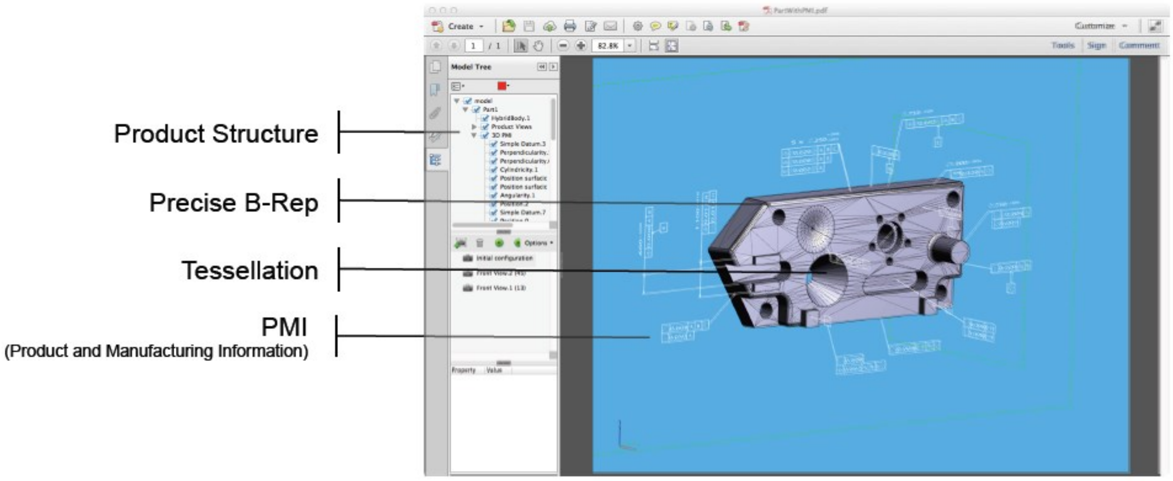


Figure 20 Contents of a 3D-PDF dataset (Spreier, 2013)

3.6.5 Parasolid XT

Parasolid itself is a geometric modeling kernel that is utilized in Siemens PLM software including NX, Teamcenter, Solid Edge and Tecnomatix. It is also utilized as a base of many independent software vendors' applications. (Siemense PLM Software, 2013) Parasolid can represent wireframe, solid surface, cellular and general non-manifold models. Parasolid XT is an open format that can be accessible by applications that do not utilize Parasolid kernel. Parasolid XT stores topological and geometric information that defines the shape of the models. (Siemens PLM Software, 2008)

3.6.6 DXF

Drawing Interchange Format (DXF) is a format generated by Autodesk AutoCAD. DXF works as an interchange format for drawings between AutoCAD and other CAD-software. DXF files can be either an ASCII or a binary format, of which ASCII format is more commonly utilized. Both of the file types provide a complete description of a drawing. A DXF file consists of a tagged data representation in which all the data elements are tagged with group codes that indicate which information the data element contains. (Autodesk, 1998) The DXF files are often utilized within sheet metal cutting, in which the 2D-geometry can be automatically attained from the DXF file by the CAM-software.

3.7 Feasibility study for two Canadian aerospace companies

Quintana et al. investigated the capabilities of two Canadian aerospace companies in order to change their operation mode towards a drawing-less environment. The study was performed in 2010 and the main objective was to investigate the possible benefits and issues that needed to be overcame in order to implement an MBD environment. The study was based on thirty-four interviews that were performed within different departments of the two companies. (Quintana, et al., 2010)

The technical section of the study was performed by adding annotations from either a detail or assembly drawing to the 3D-model. As a result, a file size reduction of 25-30 % was achieved while having only an annotated 3D-model compared to having both a 3D-model and a 2D-drawing. In addition, having a single MBD dataset, there is no need to handle separate files. It also eliminates the possibility to have deviations between the model and the drawing. However, no time savings were achieved in the study by adding the annotations to the 3D-model compared to adding them to the 2D-drawing. As a reason for that, they claimed the similarity of the 3D-annotation tools compared to the 2D-drafting tools. (Quintana, et al., 2010)

As a result of the interviews, there were several concerns regarding how an MBD model will be capable of fulfilling the applications where the 2D-drawings have been part of. List of the concerns according to Quintana et al. (2010) is presented below:

1. Data accessibility and visualization. Downstream users may not have an access to a CAD-software, which arises a need for a visualization tool
2. Data content. The downstream users must be confident that all the required data is provided through the MBD model.
3. Data presentation. The MBD model must follow an international standard in order to have data that is organized and structured.
4. Data management. An appropriate method for revising the MBD datasets is needed.

5. Data security. A method is needed to ensure the security while interacting with the MBD datasets (e.g. confidentiality, authentication)
6. Data retention. How will the data be retained while currently mainly 2D-data is stored?

In conclusion of the feasibility study, the group stated that replacing the 2D-drawings with an MBD datasets is beneficial, but some of the benefits in terms of the cost savings are not completely clear. Hence, the companies were not fully convinced in order to change their operating mode towards the drawing-less environment throughout the whole product lifecycle. Quintana et al. also concluded that many of the MBD benefits might be gathered at the manufacturing and inspection processes. (Quintana, et al., 2010)

3.8 Supplier studies of U.S. Army

U.S. army has been one of the pioneers in developing the model-based definition. They have conducted two researches for their subcontractors regarding model-based definition. The first research was conducted in 2009 and the main objective of the research was to investigate the capabilities of their suppliers in order to operate in an MBD environment. The second study was conducted in 2012, when the main objective was to gather feedback from the 3D-annotated technical data package (TDP) of a model.

3.8.1 Supplier study in 2009

In 2009, U.S Army research laboratory and Army manufacturing technology (ManTech) program gave a sponsorship to the National Institute of Standards and Technology (NIST) Manufacturing Extension Partnership (MEP) and BAE Systems Ground Systems. As a result of the sponsorship, the organizations formed a partnership in order to determine the capabilities of U.S manufacturers to operate in an MBD-environment. The study was directed to companies that operated in a supply chain that manufactures military vehicles for the U.S Department of Defense. The study was performed during year 2009, when the U.S Department of Defense was aiming to transfer the manufacturing based on 2D-drawings to the MBD-environment. (NIST MEP, 2009)

The study was conducted by a confidential electric questionnaire that was presented to the subcontractors. The questionnaire was sent to 850 companies of which 450 provided an answer. Based on the answers, the companies were divided into 5 different categories based on their capability of working as a part of Model Based Enterprise. Table 1 defines the characteristics of each class. (NIST MEP, 2009)

Table 1 Characteristics of MBE capability classes (NIST MEP, 2009)

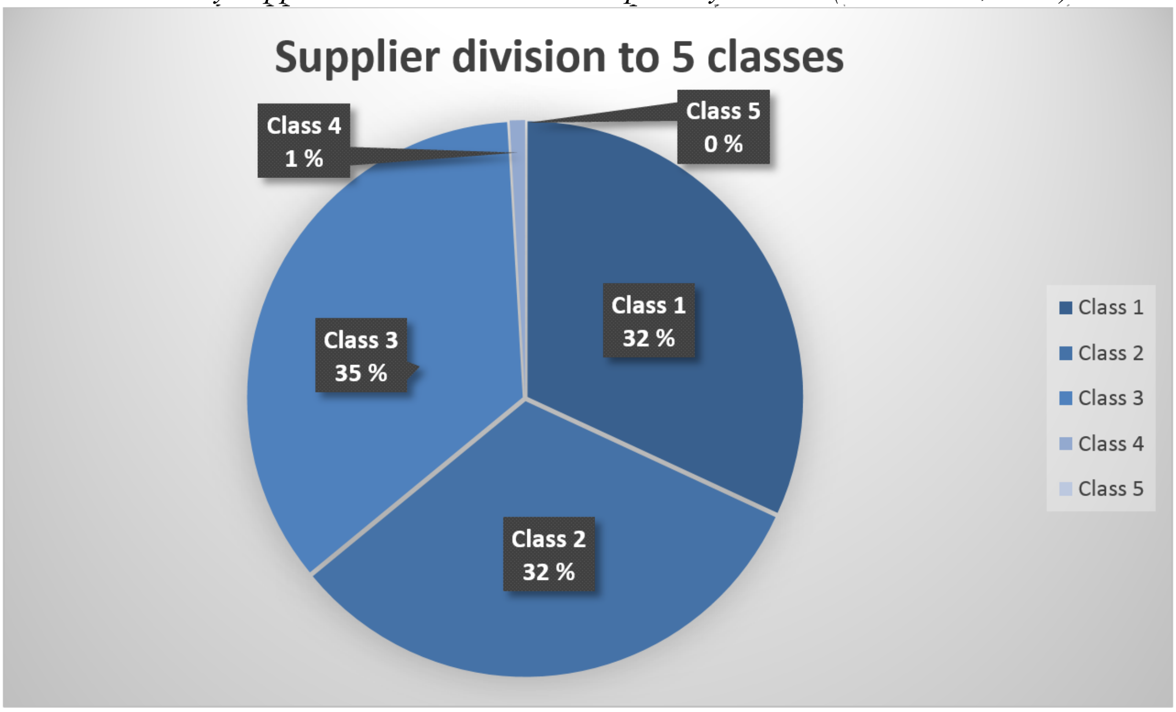
	Manufacturing process	Capability to utilize 3D data	Software used in cross department functions
Class 1	Mainly manual machines. Operations based on only 2D drawings	Very little or no	Some software utilization, but no cross-department integration or data re-utilization
Class 2	Both manual and NC machines. Operations based on mostly 2D drawings	Minor capability --> data is converted to 2D prior to manufacturing	Some amount of cross-department integration and data re-utilization
Class 3	Mostly NC machines. Utilization of 3D-models, 2D-models and 2D-drawings	Capable of utilizing 3D data --> still some usage of 2D data	ERP systems utilized in cross-department integration
Class 4	All manufacturing processes designed and programmed based on only 3D-models	All manufacturing processes designed on basis of 3D-models	Some usage of PDM/PLM
Class 5	All manufacturing processes designed and programmed based on only 3D-models	All manufacturing processes designed on basis of 3D-models	PDM/PLM operates as a dataintegration center of the company

Companies were divided to the classes as follows:

- Class 1: 142 Companies
- Class 2: 143 Companies
- Class 3: 156 Companies
- Class 4: 4 Companies
- Class 5: 0 Companies

Chart 1 determines the division of the companies into 5 categories. 66 companies of the companies that were classified to the Class 1, were located there due to incomplete answers. In conclusion, it can be said that most of the companies were classified to the classes 2 or 3 meaning that they had a moderate capability to operate in the MBE environment. (NIST MEP, 2009)

Chart 1 U.S army suppliers divided into MBE capability classes (NIST MEP, 2009)



3.8.2 Supplier study in 2012

In 2012, a research group from a company Catalyst Connection conducted a study about the usability of a 3D-TDP. The study was sponsored by US Army's Armament Research, Development & Engineering Center (ARDEC) and The National Institute of Standards and Technology (NIST). The companies interviewed in the study were from the U.S. army supply chain. The individuals contacted in the companies were mainly responsible for receiving technical data and models, handling quotes and estimations and working with the models related to the design and production. The main targets of the project were to familiarize the companies to the 3D-TDP, gather feedback about the features and identify the most usable and the most probably utilized features. (Catalyst Connection, 2012)

According to the findings, the most usable features of a 3D-TDP are the embedded native CAD- and STEP-files and a fully annotated model that is 3D-viewable. In addition, 89 % of the respondents answered that the 3D-TDP is better or much better than a 2D-drawing in order to express the design intent. The same percentage answered that the 3D-TDP contained all the information needed to manufacture a part. The most usable applications for the 3D-TDP according to the feedback were manufacturing planning, CAM-program development and expressing the design intent to the shop floor employees. (Catalyst Connection, 2012)

3.9 Conclusions of MBD literature review

The use of MBD has been increasing in recent years, which can be explained by the several benefits it offers, and by the CAD-tools that have been developed. MBD has several benefits that can be considered as facts:

- Single data source
  - Eliminates the need to handle two separate files
  - Eliminates the risk that the two files do not correspond to each other
  - Less space required to store the data

- Better visualization of the part
    - Possibility to rotate, zoom and make sections to the model
    - Possibility to measure the model
    - Highlighting the associated features by selecting a PMI-annotation

In addition to those facts, MBD is said to have other benefits including:

- Reuse of the PMI-annotations in e.g. CAM- and CMM-programming
- Decreasing the time required in the design process
- Decreasing the need of reparations to the model
- Decreasing the response time of subcontractors

The reuse of the PMI-annotations is most likely going to work, if all the software utilized are from the same company. Problems may occur, if the original file is converted to a neutral format and used in CAM- or CMM-programming. The last three benefits are such that they need to be tested. Otherwise it cannot be verified, whether the benefits can be achieved.

Currently, there are several comprehensive standards concerning MBD. The most natural choice for Konecranes, would be the international standard ISO 16792. There are also several file formats that can be related to MBD. Some of them (e.g. JT and 3D-PDF) are designed to work as a visualization of the model, while others (e.g. STEP and IGES) can be utilized in downstream operations including CAM- and CMM-programming.

The U.S. Army conducted supplier surveys in 2009 and in 2012. The first survey investigated the capability of suppliers to operate in an MBD environment, while the second survey investigated the most usable features of a 3D-TDP. As a result of the first survey, it can be said that in 2009, most of the suppliers had a moderate capability to operate in the MBD environment. The moderate capability means that they had certain capability to utilize the 3D-models and CAM, but many operations were still done with the 2D-drawings. The result of the second survey was that the most usable features of the 3D-TDP were the annotated 3D-visualizations and the embedded STEP-files.

In a feasibility study conducted to two aerospace companies (2010), the benefits of MBD were mostly seen as it being a single data source. In the study, no time saving were achieved in the design process by using MBD compared to the traditional design approach. There were also certain concerns about the MBD approach. The concerns included:

- Data availability. How can the 3D-model be opened if there is no access to CAD software?
- Data content. Downstream users must be confident that model contains all the required information.
- Data visualization. The data must be expressed in a standardized way.
- Data retention. How should the data be retained to be sure that the file can be opened after 50 years?

Currently, most of the issues above have been solved. As an example, 3D-PDF and JT-formats can be opened with free and lightweight viewers. The MBD standards also confirm that all the required information is represented in a standardized way. Especially 3D-PDF can be said to be easily retained for the next 50 years, because the 2D-drawings are also retained in the same format. JT is also an ISO standard for data visualization, meaning that it will most likely stay in use as well.

4 Mechanical design & manufacturing at Konecranes

The next chapters will give a review of the current methods within the mechanical design and manufacturing of Konecranes. The data for the chapters has been gathered by self-made surveys, interviews and observations on the factory floor. Certain written material has been utilized in order to support the own observations.

The mechanical design review has two different chapters. The first chapter will present the methods of the mechanical design in product development. The data is mostly gathered by own experiences of how the process goes and some internal material has been used as a support. The second chapter presents a survey made of design items of complete products. The survey was made by going through the 3D-models of the products in order to gather the knowledge about the parts that have been designed.

The manufacturing review is also divided into two separate sections. The first section presents three Konecranes manufacturing facilities located in Hyvinkää. The data for the first section has been mostly gathered by interviewing relevant persons and visiting the factory floors. Certain written material has also been used to support the interviews and own observations. The second section presents a subcontractor survey conducted in the autumn 2015. The survey gives a review of six Konecranes subcontractors and their ability to utilize for example 3D-models and CAM.

4.1 Introduction to Konecranes processes

At Konecranes, there are two basic processes that define the basic lines on how the production is processed after the order from a customer. The first process is called configured to order (CTO), which is used for high volume products that do not need a lot of engineering during the production. Another process type is engineered to order (ETO), which is used for the products that need special engineering including port cranes and heavy duty cranes. (Konecranes, 2011) CTO products usually have relatively high volumes and stable manufacturing chain, in which the manufacturers remain the same and the manufacturing processes are optimized. ETO products have smaller volumes and the manufacturing chain varies.

Konecranes manufacturing occurs in different locations. Typically, the most critical components, such as rope drums and gears of the heavy duty cranes, are manufactured in-house. Another approach is to do the design in-house and outsource the manufacturing to an external supplier. In addition, the Konecranes products naturally include a lot of commercial and standard parts such as screws and nuts. The different part types can be seen in Figure 21.

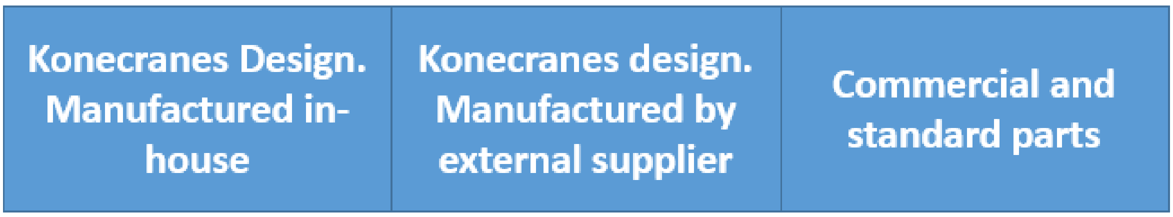


Figure 21 Different part types within Konecranes products

4.2 Current design process at product development

There are several mechanical design departments in Konecranes. Each department is responsible for its own tasks and the design methods and tools can vary depending on the department. Mechanical design takes place in departments including research, product development and product platforms. There is also a mechanical design team in India that is responsible of the modeling of mostly standard and commercial items. Despite the different design methods and tools, this thesis will concentrate on 3D-design performed at product development.

The role of the mechanical design and analyses team of product development is to develop new products and assist in developing and maintaining the existing products. Usually the product development requests are originated from either research department or product management. The basic tools of the department are NX and Teamcenter, which are both from the company Siemens PLM and software. NX is the CAD-software whereas Teamcenter is the Product lifetime management (PLM) software. The software are well integrated, because they are from the same producer and designed to operate together. Konecranes also has other CAD-software (e.g. SolidWorks, Vertex and AutoCAD) and other PDM-systems (e.g. ATON). However, those systems are not significantly utilized within product development.

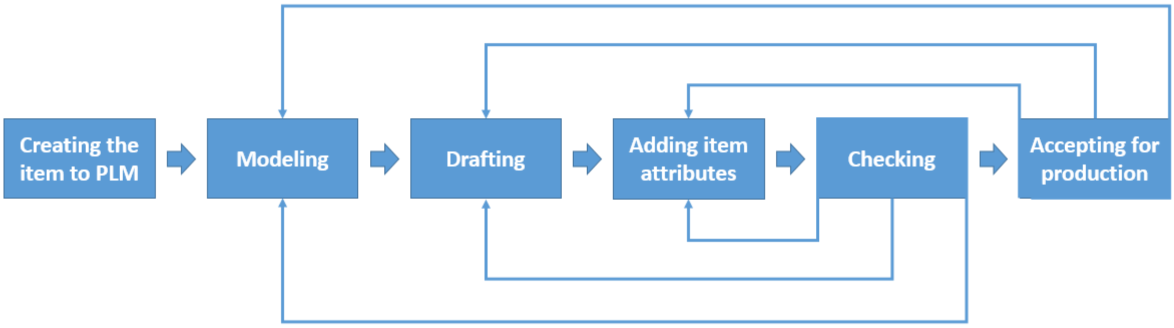


Figure 22 Current process flow from software point of view

Figure 22 determines the basic mechanical design process flow from the software point of view. Within the PLM, the parts are called items. Before the usage of NX and Teamcenter, the designer usually has some sketches and ideas about the design of the product. The first step of the process that utilizes the software, is to create the item to the PLM-software (Teamcenter). Here, the item receives its own identification number (ID). All the datasets (3D-model, 2D-drawing etc.) of the certain product are later created under the same item ID. After the item creation is complete, the next step is to make a 3D-model of it by using NX. A 2D-drawing is created from the 3D-model after the modeling phase is finished. The drawing contains all the information, such as dimension and tolerances, needed in order to manufacture the part. After the model and the drawing are ready, item attributes, such as description and keywords, are added for the ID. The attributes are also mostly included in the 2D-drawing.

After the item attribute addition is complete, the design is transferred to Teamcenter workflows. There are several workflows, but from the mechanical designer's point of view the most often utilized workflows are Check Item Revision and Accept for Production. The idea of the both workflows is to verify that the model, the drawing and the attributes are made correctly and the item is ready to be manufactured. As Figure 22 shows, the process is iterative meaning that changes to the model can be applied before the checking or accepting for production. Figure 23 presents the product lifecycle from the beginning to the end including the statuses available in PLM-system.

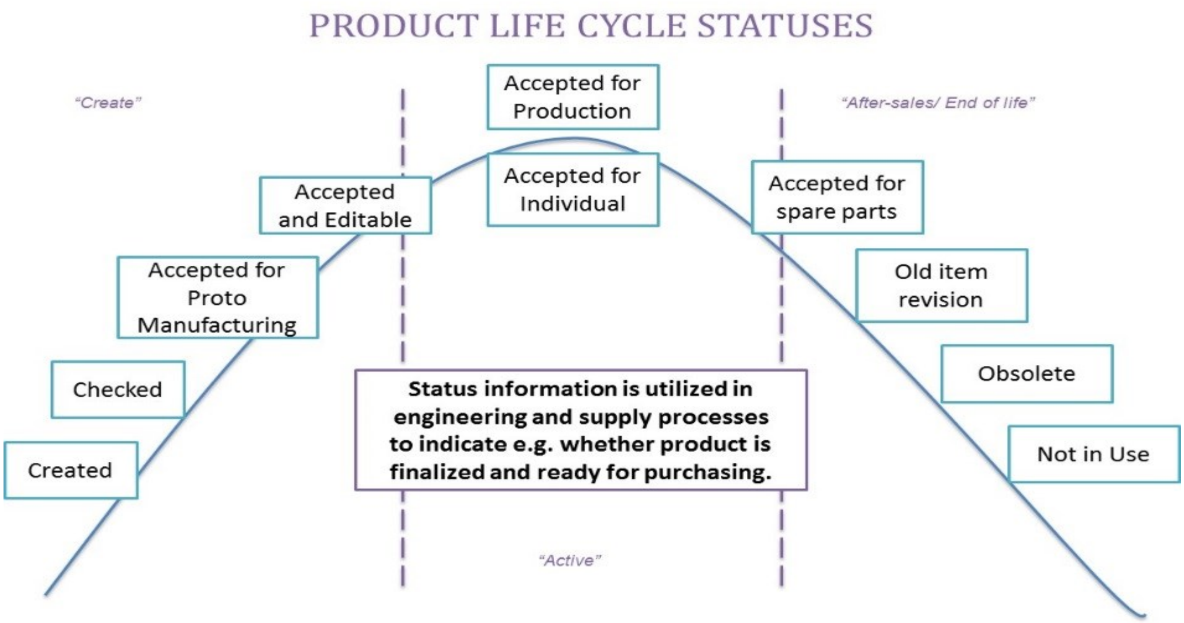


Figure 23 Product lifecycle statuses (Laukkanen, 2015)

Figure 24 shows the datasets of a basic sheet metal plate in Teamcenter. The item contains a 3D-model, a 2D-PDF, a DXF-file for cutting and a drawing model. The 3D-model and the drawing-model are integrated meaning that the drawing updates automatically if the model is edited. The creation of the 2D-PDF and the DXF-file needs to be repeated after any changes are applied to the model to achieve up to date files. All the items belonging to an assembly are listed under the folder "View". In this example the item is not an assembly meaning that there is only a material item linked to the design item. The checkered flag after the dataset denotes that the dataset is accepted for production and cannot be edited.

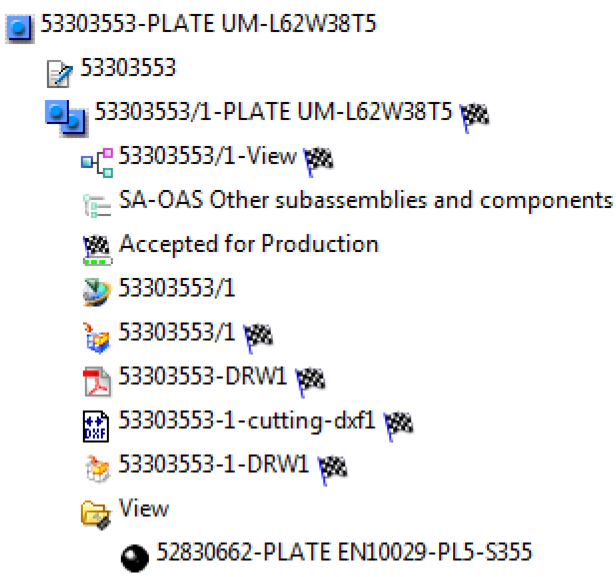


Figure 24 Example of datasets of sheet metal plate

Currently, the most typical document that is delivered further to the manufacturing is a 2D-drawing in PDF-format. In case the item requires sheet metal cutting, a cutting DXF is also delivered. Teamcenter supports the conversion of NX-models to Parasolid, STEP and 3D-PDF. The conversion of an NX-model to a DXF is also supported, if the model contains a flat pattern view of a sheet metal. In addition, NX-drawings can be converted to a DXF or DWG formats. NX also creates a JT-file automatically of each 3D-model.

4.3 Design item survey of existing products

To determine what kind of parts the Konecranes products contain, a design item study was conducted. The study was done by going through each individual design item and assembly of a hoist and a hoisting trolley. Standard items and commercial items, such as screws, nuts and limit switches were not taken into account. Sections that were investigated in each item were:

- Item ID
- Item Description
- Manufacturing method
- File formats
- Material
- Weight

Some of the items and assemblies existed multiple times in the main assembly, while others existed just once.

The first study was made for a hoisting trolley called UM. UM is a heavy duty overhead crane that can be utilized in an assembly or a process use. Smarton is a model name for UM trolleys that are sold under Konecranes brand. The lifting capacities of the UM trolleys range from 6.3 tons up to 250 tons with a single trolley and up to 500 tons with two trolleys. The volumes of UM are relatively low compared to smaller hoists. The study of design items included in a UM trolley was conducted by going through each design item in a 3D-model of a mid-range UM trolley. Figure 25 presents the 3D-model of the UM-trolley.

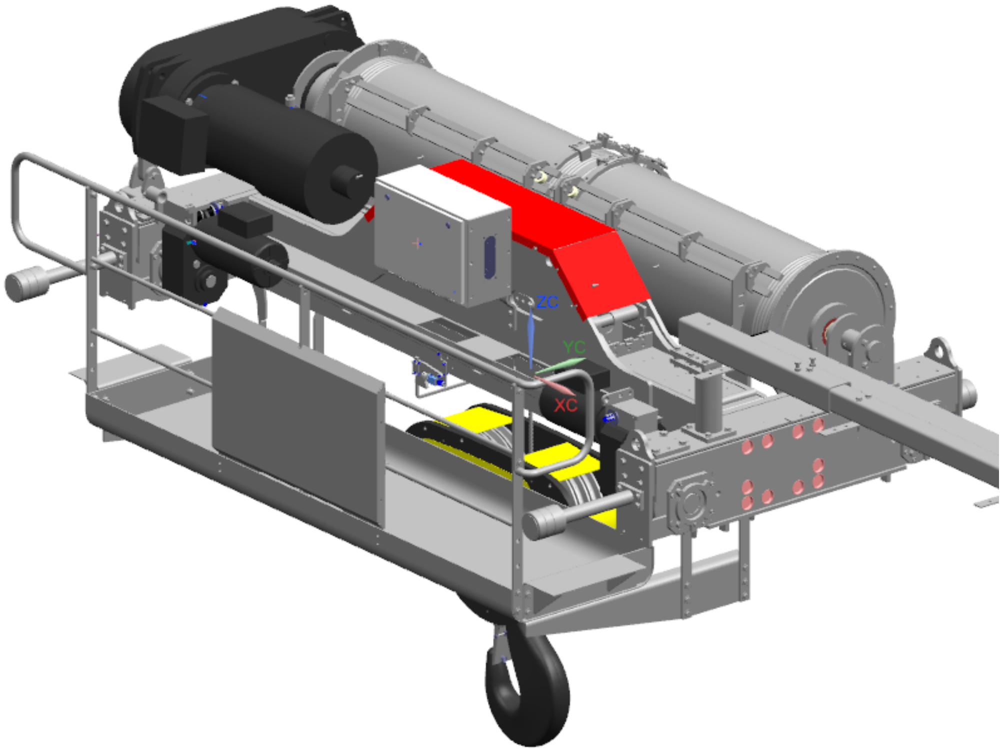


Figure 25 3D-model of the UM-Trolley

The second study was made for a smaller hoist called Q-hoist. It is a wire rope hoist that is made for assembly workshops and industrial plants. CXT is the name of the Q-hoist model sold under Konecranes brand. The lifting capacity of the Q-hoist ranges up to 80 tons. The Q-hoists are high volume products that are sold several thousand each year. The study was conducted by going through the 3D-model of a mid-range Q-hoist with a low-headroom structure, which is one of the most typical models of the hoist. The Q-hoist has been on the markets about fifteen years and certain amount of changes has been done to the design during the years. Figure 26 presents the 3D-model of the Q-hoist.

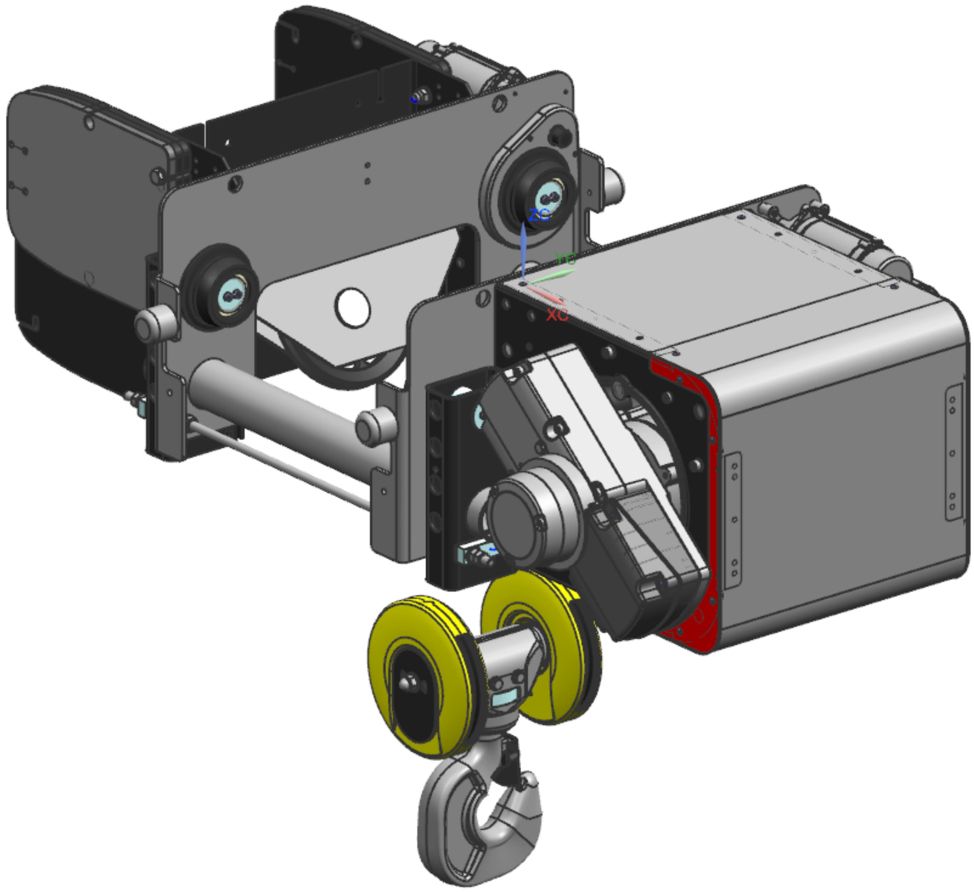


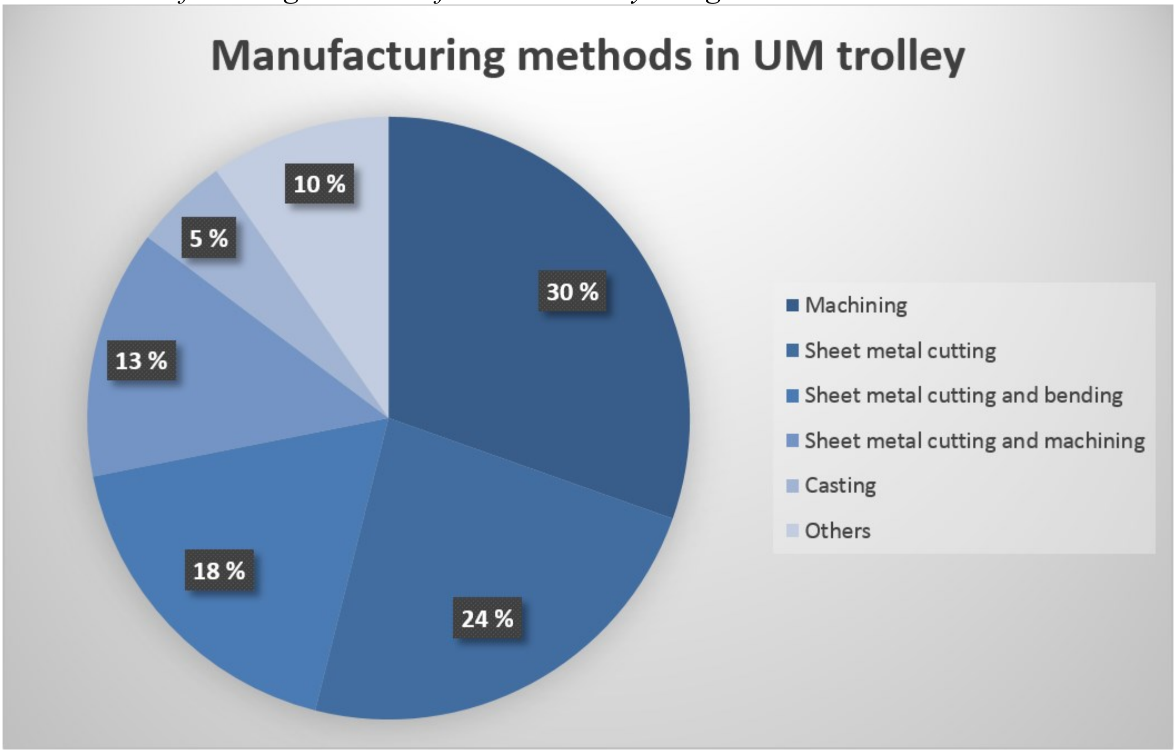
Figure 26 3D-model of the Q-hoist

4.3.1 Results of UM-trolley

The design of the items in a UM-trolley has been performed in various design departments. The oldest items are taken from the old models while the majority of the design work is made about five years ago. Hence, there are differences in the design itself as well as in the file formats and the manner of representation of the design. Most of the items in a UM-trolley do have a 3D-model. However, certain items contain only visual models of the covers and do not correspond accurately to the 2D-drawings.

77 % of the total amount of the design items were individual items and 23 % were assemblies. The distribution of the manufacturing methods is presented in Chart 2. All the metal plates regardless of the thickness of the plate are called sheet metals.

Chart 2 Manufacturing methods of the UM-trolley design items



The manufacturing method "machining" contains all the items manufactured by milling, turning and boring. Whether the machining is done for a cut plate, the manufacturing method is in a separate method: sheet metal cutting and machining. The assembly items can be divided into two separate assembly methods:

- Assembly: 54 %
- Welding assembly: 46 %

A basic assembly is typically an assembly that is done by bolt joints and a welding assembly is typically a joining of sheet metal parts.

Each of the design items contained at least a 2D-drawing in PDF-form, which is the basis of the manufacturing information in each item. 90 % of the items contained also a 3D-model. The distribution of the other file formats is as follows:

- Contains a DXF or a DWG: 45 %
- Contains calculations: 8 %
- Contains a VXP (Vertex drawing): 8 %
- Contains a 3D-PDF: 0.3 %

The total percentage of the sheet metal parts in the trolley is 56 %. The cutting DXF is an important file in a sheet metal part, because the cutting geometries are usually generated automatically based on that. Of the total amount of sheet metal items, 66 % contained a cutting DXF. This can be determined to be a first step of MBD, in which the geometry is defined by a model instead of a drawing. The sheet metal thicknesses were divided according to the Konecranes classification:

- Thin sheet metals (0-3 mm): 27 %
- Thick sheet metals (thicker than 3 mm): 73 %

The three most common manufacturing methods of the sheet metal items were mentioned previously. The most typical manufacturing method is the cutting which is relatively simple method and can be easily done with MBD using a cutting DXF. 32 % of the sheet metal items included metal bending which is typical for thin and medium thick sheet metals. 24 % of the sheet metal items included machining which was typically milling, boring or threading.

From the total amount of individual items, 46 % contained machining. Those items include all the parts that included metal removal except sawing and cutting. The 46 % can be divided further as follows:

- Only machining: 47 %
- Machining of a sheet metal item 29 %
- Machining of a casting 10 %

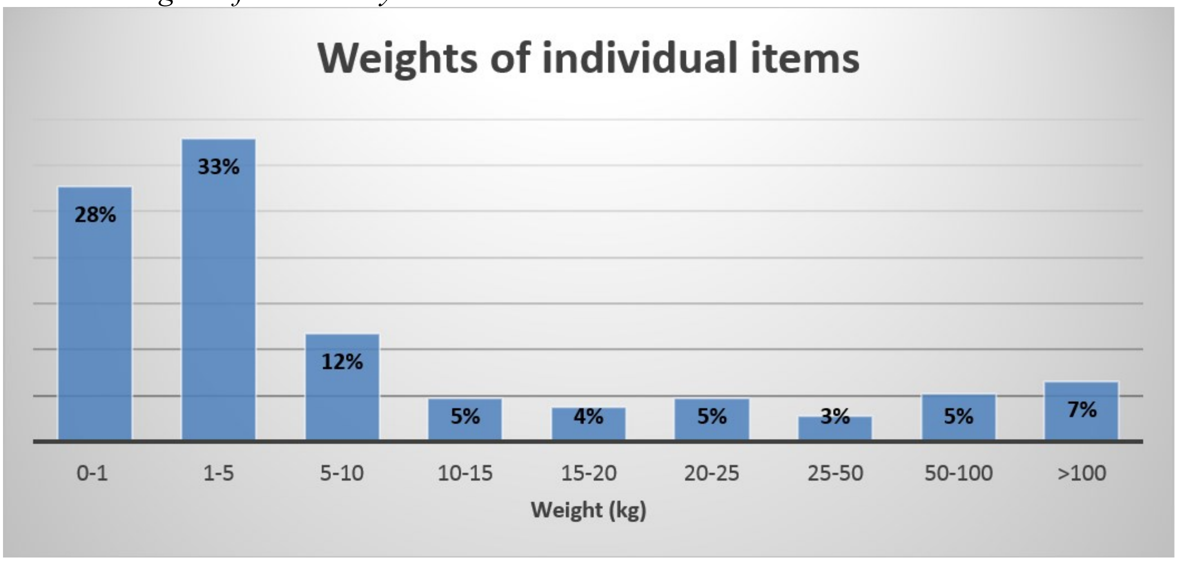
The most typical machining billets were:

- Round Bar: 35 %
- Sheet Metal or Flat Bar: 33 %
- Casting: 10 %
- Round Tube: 7 %

During the study it was determined that certain design items only contain a visual 3D-model of the cover. Other details of those items are listed below:

- Contains DXF or DWG: 81 %
- Machining items: 85 %
    - Turning: 52 %
    - Milling: 48 %
- Casting items: 15 %
- Sheet metal items: 4 %

Chart 3 Weights of UM trolley individual items



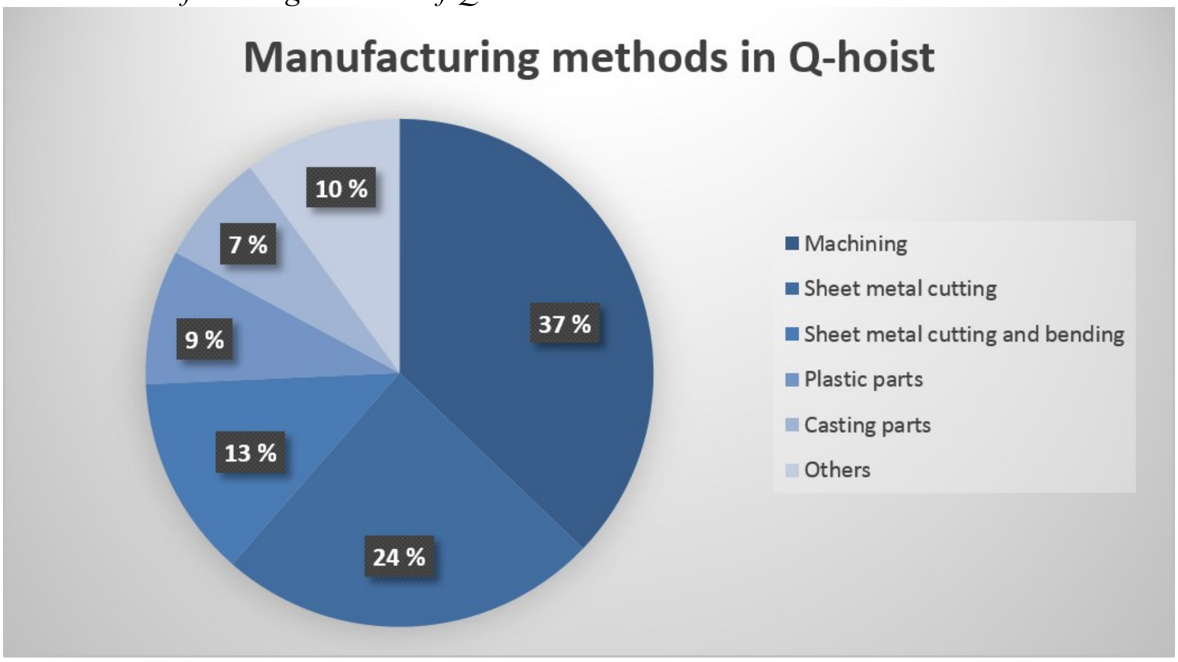
As a part of the study, the weights of the individual items were determined. The parts that were selected to this study were those that contained some kind of manufacturing and were not assemblies. Hence, for example a machining of an assembly (e.g. milling of a welding assembly) was not selected. Chart 3 shows how the percentages of the individual items are divided based on their weight. From the chart it can be seen that 29 % of the items weigh less than one kilogram and 64 % of them weigh less than five kilograms. The average weight of a single item in the UM trolley is 25.1 kilograms, but the average is raised significantly by several items that weigh more than 100 kilograms. A more descriptive indicator of the weight in this case is median weight, which is 2.7 kilograms.

4.3.2 Results of Q-hoist

The original design of the Q-hoist was made in 2D, but currently most of the items do have a 3D-model. Some of the 3D-models were done afterwards based on the 2D-drawings, and others were done when the part was updated to a newer design. However, certain items of the Q-hoist had only visual models of the covers.

The total number of design items in the 3D-model of the Q-hoist was about 34 % compared to the total number of design items in the UM-trolley. The percentage of individual items was 73 % and the percentage of assemblies was 27 %. All of the assemblies in this Q-hoist were basic assemblies with no welding. The reason for that might be that the sheet metal thicknesses are thinner compared to the UM, which allows the use of bending instead of welding two plates together. The distribution of the manufacturing methods in the Q-hoist design items is presented in Chart 4.

Chart 4 Manufacturing methods of Q-hoist items



The sheet metal items were typical in the Q-hoist as well. The percentage of them was 31 % of the total number. Only 43 % of all the sheet metal items contained a cutting DXF, which is essential for the manufacturing. The thicknesses of the sheet metals were divided according to the Konecranes classification:

- Thin sheet metal (0-3 mm): 40 %
- Thick sheet metal (more than 3 mm): 60 %

In this case the manufacturing methods of the sheet metal items were divided as follows:

- Only cutting: 57 %
- Cutting and bending: 30 %
- Sheet metal forming: 10 %
- Contains machining: 3 %

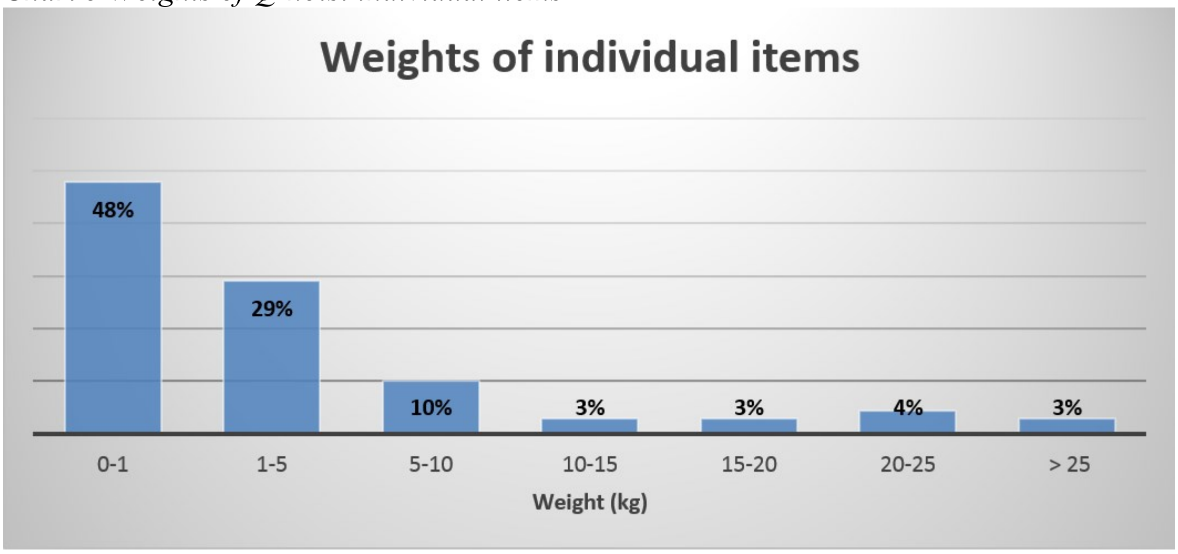
Here as well, the most typical manufacturing method of the sheet metal items was the cutting with no additional phases. The items that contain both cutting and bending were about as typical as they were in the UM-trolley. However, the sheet metal items that contain machining were very rare. The reason is most likely that the sheet metal thicknesses were significantly thinner, which makes the machining more difficult and unnecessary. The sheet metal forming (e.g. punching) was also utilized in few items.

37 % of the individual items in the Q-hoist contained machining. Those parts were divided further as follows:

- Only machining: 84.5 %
- Machining of a cast: 11.5 %
- Machining of a sheet metal: 4 %

The basic machining parts were mainly manufactured from round bars (45 %) and round tubes (36 %). Hence, it can be concluded that the majority of the machining items were manufactured by turning.

Chart 5 Weights of Q-hoist individual items



As a part of the study, the weights of the individual items were determined. The items were selected to the weight study similarly as with the UM-trolley. Chart 5 shows how the percentages of the different items are distributed based on their weight. From the percentages in the pillars, it can be seen that 48 % of the design items in the Q-hoist weigh less than one kilogram and 77 % of them weigh less than 5 kilograms. The average weight of the items is 4.6 kilograms, which is raised significantly by the heaviest items. A more descriptive indicator in this case is the median weight, which is 1.4 kilograms.

4.3.3 Conclusions of the design item survey

In conclusion of the study it can be said that the most typical manufacturing methods among the products are machining and sheet metal manufacturing. In more details, turning and sheet metal cutting with no extra work phases are the most common methods. Concerning the sheet metal items, it is especially important to include a cutting DXF for the item, because it is needed to create the cutting tracks. 66 % of the UM-trolley sheet metal items included a cutting DXF and only 43 % of the Q-hoist sheet metal items included it. The differences between the UM-trolley and the Q-hoist sheet metal items were in the sheet metal thicknesses, and in the way the assemblies were generated. The UM-trolley contained a lot of welding assemblies, which is most likely because the thick plates of it are not possible to be bent. In addition, the structure contains a lot of elements that are kind of beams and contain a lot of sheet metal structures. The Q-hoist did not have any welding assemblies, but more bent sheet metal items.

Based on the study it can be said that it is important to have a proper 3D-model of each item of the product. The visual models are better than no models at all, but for example assembly instructions can be made a lot better with less time, when utilizing proper models. In addition, the proper models of the machining items can be utilized in the creation of a CAM-program. Another recommendation is that each part that contains sheet metal cutting, should include a cutting DXF. If it does not exist, it needs to be generated by the manufacturer, which is an extra cost and the generated DXF does not probably correspond to the original design.

4.4 Konecranes manufacturing facilities at Hyvinkää

The manufacturing facilities discussed in the following chapters include rope drum, gear and spare part manufacturing. The data for the rope drum and gear manufacturing is gathered by an interview, a factory visit and a written report. The data for the spare part manufacturing is gathered by an interview of the factory manager and a visit to the factory.

Konecranes manufacturing facilities in Hyvinkää are able to utilize three different CAM-software: PEPS, WinCam and SolidCAM. PEPS is a software that is utilized for creating the machining paths for the turning of the rope drums. The current PEPS software used at the factories is not compatible with Windows 7, which causes certain problems. In addition, there is no technical support for the version in Finland anymore. (Heinonen & Marjosalmi, 2012)

WinCAM is a CAM-software that is developed in Finland. WinCAM is basically developed for turning and has limitations in milling operations. In addition, WinCAM operates based on only 2D-models, so it is not an optimal solution for the efficient utilization of the 3D-models. (Heinonen & Marjosalmi, 2012)

SolidCAM is a software that is developed to be integrated with SolidWorks. SolidCAM includes various modules for different machining actions including 3D milling, simultaneous 5-axis milling and turning. (SolidWorks, 2015) Konecranes factories have one license of SolidCAM 2010 for milling. In addition, SolidCam 2014 is utilized in Konecranes U.S. and the same licenses can also be utilized in Finland due to the time differences. The newer version of SolidCAM can be used for both milling and turning.

4.4.1 Rope drum & gear manufacturing

The rope drum and gear manufacturing of Konecranes is performed mainly at Hyvinkää. Typical products manufactured at these factories are rope drums, gear boxes, shafts and gearwheels. Heinonen & Marjosalmi performed a study (2012) as an industry project for Aalto University. The project objective was to find a replacing CAM-software for PEPS due to the issues mentioned in the previous section. In conclusions of the study, NX-CAM was selected to be the best solution, because it would have the best compatibility with the NX-CAD that is utilized within design departments. However, NX-CAM was abandoned due to the high price and the selected CAM-software was WinCam. (Hokkanen, 2015)

The current situation at the rope drum manufacturing is that WinCam is utilized in certain operations, but the creation of the rope grooves is still done with PEPS. The reason for that is that WinCam is not able to create the rope grooves as well as PEPS. SolidCAM is also utilized in certain operations, and a target is that the usage of SolidCAM should be increased. (Hokkanen, 2015) The workflow at the rope drum manufacturing goes so that the NC-programmer receives a 2D-drawing and the profile to be turned is drafted to the PEPS-software based on the drawing. When a new part arrives to the production, the programmer determines, whether there is a part that has a geometry close to the new part. Typically, the parts are quite similar, and an older program can be used as a template. Most of the parameters can be inserted directly to the macros of PEPS, which makes the programming easier by removing the need to completely re-draw the profile of the drum. (Heinonen & Marjosalmi, 2012) Figure 27 shows the proceeding of the program creation in the rope drum manufacturing.

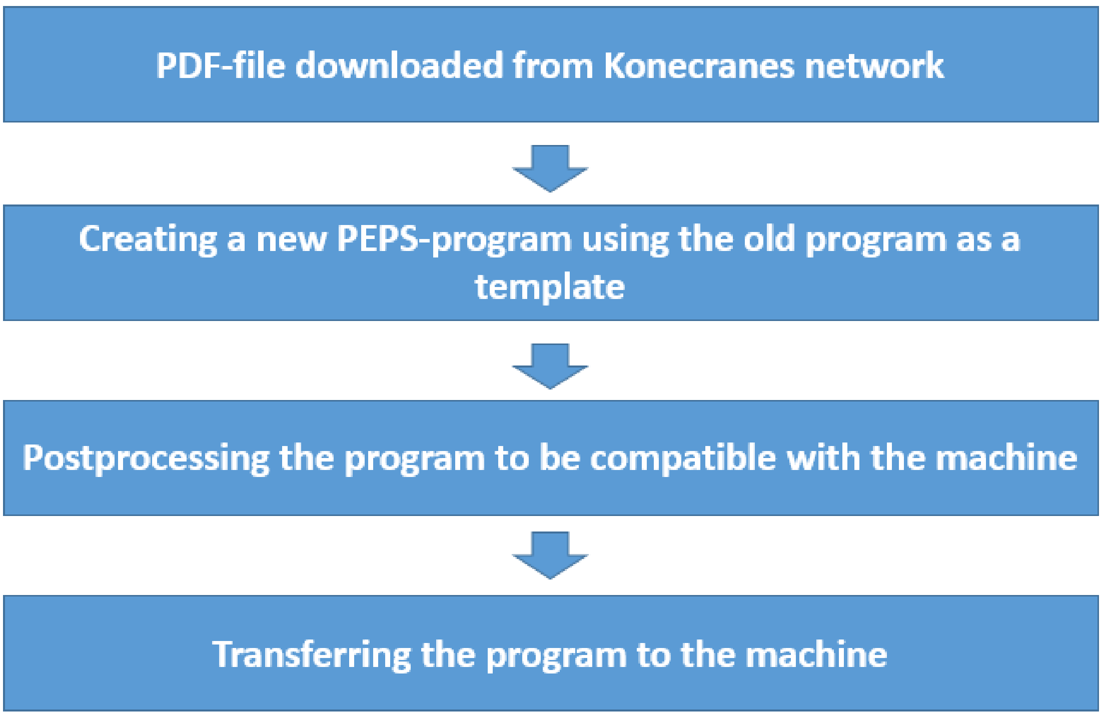


Figure 27 NC-programming process at rope drum manufacturing according to Heinonen & Marjosalmi

The parts manufactured by the Hyvinkää gear factory are gear wheels, shafts and gear boxes. The CAM-software used at the gear manufacturing is WinCam and it is utilized in turning operations. The CAM-programs are built manually based on 2D-drawings. Many parts manufactured are rather similar, so existing programs can often be utilized by only changing certain parameter values in the macros. The milling programs of the gearboxes and gear wheels are usually generated manually using macros and the controllers of the machines. (Heinonen & Marjosalmi, 2012)

The machining time of a single part varies from few minutes to three hours, and the batch sizes vary from one to ten. Due to the small batch sizes, the time spent in the programming is especially critical. The time spent in the machining of a basic part is usually from 15 minutes to one hour. If the part is simple and there is an existing program for almost similar part to be used as a template, the programming can be done in 15 minutes. If the part is completely new, but quite basic otherwise, the programming time required is about one hour. For programming the most complex part the time needed can rise to few hours. (Heinonen & Marjosalmi, 2012)

4.4.2 Spare part manufacturing

Konecranes manufactures spare parts at Parts Manufacturing Centre (PMC) in Hyvinkää. The spare parts are manufactured for own products, competitor products and machine tools. The spare parts can be manufactured for both old and new equipment, but the majority of the parts are made for old products. There are certain parts among the spare parts that are manufactured to stock, but typically they are manufactured based on orders. A typical batch size is relatively small. Most of the customers of PMC are internal, but sometimes the orders can originate from external customers. PMC also does assembly work for parts certain that can be manufactured either at PMC or somewhere else. (Mäkelä, 2016)

The manufacturing process at PMC depends on the part to be manufactured. In case of a basic part that is manufactured frequently the process is quite simple, because the NC-programs etc. are already existing. However, sometimes the drawing of a part can be very elementary or even not existing; Reverse engineering can be utilized in case the part itself exists already. The NC-programs are usually generated by CAM-software (SolidCAM or PEPS), but they can also be made manually or by interactive workshop programming at the machine. Each machinist has an own computer next to the machine tool, which gives the possibility to check the virtual documents from the computer and makes the paperless environment possible. (Mäkelä, 2016)

The machines used at PMC are both manually and numerically controlled. The majority of the machines are lathes and milling machines, but PMC also has a broaching machine, grinders and welding machines. The NC-machines are typically used when the batch is larger than just few pieces or the part is too complex to be manufactured with a manual machine. The manual machines are used when the batch is small or the NC-machines cannot do the machining due to a certain reason (e.g. size). The software, that PMC can utilize, are listed below:

- SolidCam
- SolidWorks
- Teamcenter
- Alfresco
- PEPS (when NC-programming is made for rope drums)

SolidWorks can be utilized for CAD-modeling and model inspections. SolidCAM can be utilized in order to create the NC-programs and simulating the manufacturing. Teamcenter and Alfresco are utilized to handle the documents. (Mäkelä, 2016)

4.4.3 Conclusions of Konecranes manufacturing

The rope drum and gear factories mainly use PEPS and WinCAM in their turning programming. PEPS is used to make the programs for turning of the rope drums and WinCAM is used for other turned parts including shafts. The both programs utilize 2D-wire models that are created based on 2D-drawings. The creation of certain 2D-models has been automated by macros. The milling programs of the factories (e.g. gear boxes) are currently created mainly manually, even though SolidCAM 2010 and SolidCAM 2014 are available for the milling programming.

The spare part manufacturing uses both manual- and NC-controlled machines within their production. The NC-programming can be done by any of the three ways introduced previously: automatically, manually or interactively by the machine. For the automatic programming, PMC can utilize either SolidCAM or PEPS. SolidCAM is typically used for programming the parts to be milled and PEPS is used for programming of the spare part rope drums.

In conclusion of the Konecranes manufacturing facilities in Hyvinkää, it can be said that the capability to utilize 3D-models is rather good. If classifying the factories to the classes used in the U.S. army supplier researches, each factory would be ranked to the class 3. The class 3 means basically that the factory can utilize 3D-models and CAM, but certain actions are still made with 2D-drawings. The factories also have computers adjacent to the machine tools, which enables the implementation of the paperless environment. Concerning the rope drum and gear manufacturing, the 2D-approach is working quite well, because the major amount of parts are turned and 2D works quite nicely with turning. In addition, the parts are quite similar and the existing programs and macros can typically be used as a template. However, SolidCAM could be used to do the programming with 3D-models. Many of the benefits from 3D-models and MBD could be applied to the spare part manufacturing. There, the manufactured parts can vary a lot and certain parts are quite complex, which increases the benefit of the automatic programming and the good visualization.

4.5 Subcontractor survey – manufacturing processes

A major amount of the Konecranes parts are manufactured by subcontractors. While thinking about the implementation of MBD, it is important to define the capabilities of different subcontractors by clarifying the manufacturing processes and systems utilized. Hence, a survey of subcontractors was conducted during the autumn 2015.

The manufacturers were selected based on discussions with the thesis advisor and a sourcing manager of Konecranes. The manufacturers selected for the survey are significant suppliers located both in Finland and abroad. The survey itself was conducted by contacting the supplier representatives with email. The contact information of the representatives were gathered from the supplier managers that handle the co-operation between Konecranes and the supplier. To maintain the anonymity of the companies, they are called Supplier A, Supplier B etc.

The questions asked in the survey are listed below:

- Which are the main Konecranes products that are manufactured at your factory?
- What manufacturing information (engineering drawing, 3D-model, etc.) do you get from Konecranes and how the information is utilized?
    - E.g. 2D engineering drawing, which is utilized in order to manually make an NC-program?
- Which machines are used in manufacturing?
    - Manual / NC?
    - How the machines are controlled and how the programming is done?
- Possible problems and development ideas?
    - In documents and models?
    - In process?
- How is the quality control performed?
- Which data formats can you utilize (STEP, IGES, DXF, etc.)?
- How is 3D-data (e.g. STEP-model) utilized or how could it be utilized?
    - Checking the geometry or dimensions?
    - Automated NC-programming?
    - Quality control?
- How are 2D-drawings used?
- What information 2D-drawing needs to contain, if there is also a 3D-model of the item (dimensions can be checked from the model)?

4.5.1 Results from Supplier A

Supplier A is a Finnish workshop, which mainly manufactures sheet metal parts and mechanical assemblies for Konecranes. The volumes of the parts vary from individual prototypes to large stock batches. The file formats, which can be utilized at the supplier are DXF, DWG and STEP. According to the supplier's opinion, the benefits of the 3D-models can mostly be seen when the 2D-drawing is insufficient or certain dimensions need to be confirmed. Even though certain machines can utilize 3D-models, Supplier A does not have a large benefit of them because sheet metal cutting is made based on 2D-models (DXF). The requirements for a 2D-drawing are that they should include a sufficient amount of dimensions and aim to visualize the geometry as much as possible. They state that the reading skills of the 2D-drawings are becoming weaker also among professionals, because the 3D-visualizations are getting more popular. (Supplier A, 2015)

The process of the sheet metal cutting and bending generally begins so that the supplier receives a 2D-drawing and a DXF-file. The 2D-drawing includes the dimensioning and other manufacturing information such as color codes. The DXF-file is utilized to create the cutting geometry for a laser cutter and strain values and edging lines for an edging machine. The machines that Supplier A uses, are only NC-controlled. The laser cutters are programmed by a Bysoft-software from Bystronic and the edging presses are pre-programmed by a Bendcam-software by Amada. (Supplier A, 2015)

The supplier representative also suggested certain development targets for the communication between Konecranes and the supplier. The first target is that the software and the drawing templates should be harmonized. The supplier faces large quality differences in the drawings and some of the drawings are still made straight to 2D, instead of generating the drawing from a 3D-model. (Supplier A, 2015)

4.5.2 Results from Supplier B

Supplier B is a Finnish workshop that mainly manufactures sheet metal parts and steel structures for Konecranes. The files that Supplier B usually receives from Konecranes are a 2D-PDF, a DXF and sometimes a DWG. The supplier also receives a Parasolid-model of all the heavy steel structures. 2D-drawings are utilized as appendixes of the work orders and other manufacturing documents throughout the manufacturing chain. The Parasolid-, DWG- and PDF-files can also be utilized in order to create a DXF-file, if the customer has not delivered it. The DXF-file is later utilized in the creation of NC-programs. In addition to the previous file formats, Supplier B can also utilize IGES-files. (Supplier B, 2015)

3D-models can be utilized in the creation of NC-programs for a machining center. The requirements for a basic 2D-drawing according to Supplier B are that they should contain all the functional information, material information, billet dimensions, bending dimensions, welding annotations and surface treatment. Supplier B mainly utilizes NC-machines in their manufacturing. The machines available at the supplier are sheet metal work centers, laser cutters, plasma cutters, gas cutters, lathes and machining centers. The quality control is implemented manually by taking the dimensions from the 2D-drawing and measuring completely the first and last part of a batch and taking sample measurements from intermediate parts. (Supplier B, 2015)

Supplier B representative also gave few problems and development ideas for Konecranes. The first problem is that the 2D-drawings can be insufficient lacking for example welding annotations or surface finish information. Another problem is that the flat pattern view of a sheet metal part is lacking. A development idea is that all the changes of the design should more reliably be applied to all the files including the DXFs. Another development idea is that the 2D-drawings of the parts should be delivered as separate files and not linked to the BOM or under the main assembly drawing. (Supplier B, 2015)

4.5.3 Results from Supplier C

Supplier C is also a Finnish workshop. The Konecranes products they manufacture are sheet metal parts and metal structures. The basic data that Supplier C receives from Konecranes is a 2D-drawing and sometimes a DXF-model. The 2D-drawings are often utilized in the creation of the DXF-model, if that does not exist already. Supplier C utilizes NC-controlled edging presses, sheet metal work centers and laser cutters. Welding is done manually. (Supplier C, 2015)

The file formats that supplier C can utilize are DXF, DWG and PDF. 3D-models cannot be utilized with current software. However, the supplier representative stated that the 3D-models, which can be examined with free software such as Adobe Reader would be useful in the manufacturing. A wish from the Supplier C is that the drawings should be delivered in the DXF or DWG format, so the supplier does not have to generate the cutting DXFs them self. (Supplier C, 2015)

4.5.4 Results from Supplier D

Supplier D in an Estonian workshop that mainly supplies heavy steel structures. The data that supplier D receives from Konecranes is typically a 2D-drawing in PDF format and sometimes a DWG- or a DXF- file. If the DWG/DXF is lacking, supplier D must make it them self. The DWG- and DXF-models are utilized in order to generate gas- and laser cutting programs and NC-programs. The programs are usually made manually, but in case the structure is complex, Mastercam is utilized. Supplier D has both manual- and NC-machines, but the manufacturing is performed mainly with the NC-machines. (Supplier D, 2015)

The file formats that Supplier D is capable of utilizing are STEP, IGES, DXF/DWG and Parasolid. They have licenses for few CAD-software including SolidWorks and Solid Edge. 3D-models can be utilized in the automatic NC-programming as well as for the geometry checking and the quality control. The quality control process is implemented so that each employee checks his own work with certified measurement instruments and a quality specialist confirms that each process is done correctly. (Supplier D, 2015)

Supplier D also gave few development ideas and problems that are encountered during the manufacturing. The biggest problem in their opinion is that the 2D-drawing is insufficient either by lacking dimensions, or by having conflicting dimensions. Another problem with the drawings is that the PDF drawing and the DXF- or DWG-model are not equivalent. A wish from the supplier was that in case the structure to be manufactured is complex, there should always be a 3D-model available. Their opinion is that all the manufacturing information should be presented in the 2D-drawing, and the model (3D or 2D DXF/DWG) should support the drawing. (Supplier D, 2015)

4.5.5 Results from Supplier E

Supplier E is a Chinese manufacturer that supplies casting and machining parts. Typically, Konecranes delivers the manufacturing information in a 2D-drawing, but 3D-models are also provided if needed. Supplier E has only NC-machines and the NC-programming can be done either manually based on the 2D-drawing, or automatically utilizing the 3D-model. Sometimes, if the 3D-model does not exist, the supplier generates it based on the 2D-drawing. The 3D-model can be then utilized in the generation of the NC-program. The 2D-drawings are utilized also in the mold creation and tooling. The 3D-models can help alongside with the drawing if the geometry is complex or the dimensions are not clear. (Supplier E, 2015)

Supplier E is capable of utilizing 2D-PDF drawings and 3D-models generated by NX. The process between Konecranes and supplier E works generally well, but sometimes the designs are too complex to be manufactured or the tolerances are too tight. The quality control is performed manually based on the instructions during the machining process. In case supplier E would receive an annotated 3D-model, they could do the NC-programs automatically and check the dimensions etc. straight from the model. If certain information would be missing from the model (e.g. special treatment or surface roughness), it should be represented in the 2D-drawing. (Supplier E, 2015)

4.5.6 Results from Supplier F

Supplier F is a Slovakian workshop that mainly manufactures sheet metal parts and steel structures. Supplier F utilizes both manual- and NC-machines in their production. The NC-machines are programmed using CAM. Konecranes typically delivers only a 2D-PDF for the supplier. The PDF is typically utilized to create a 3D-model, which is utilized in the CAM-software in order to create an NC-program. One clear development idea from the supplier is that Konecranes should deliver the 3D-model straight to them to avoid extra work caused by the modeling. (Supplier F, 2015)

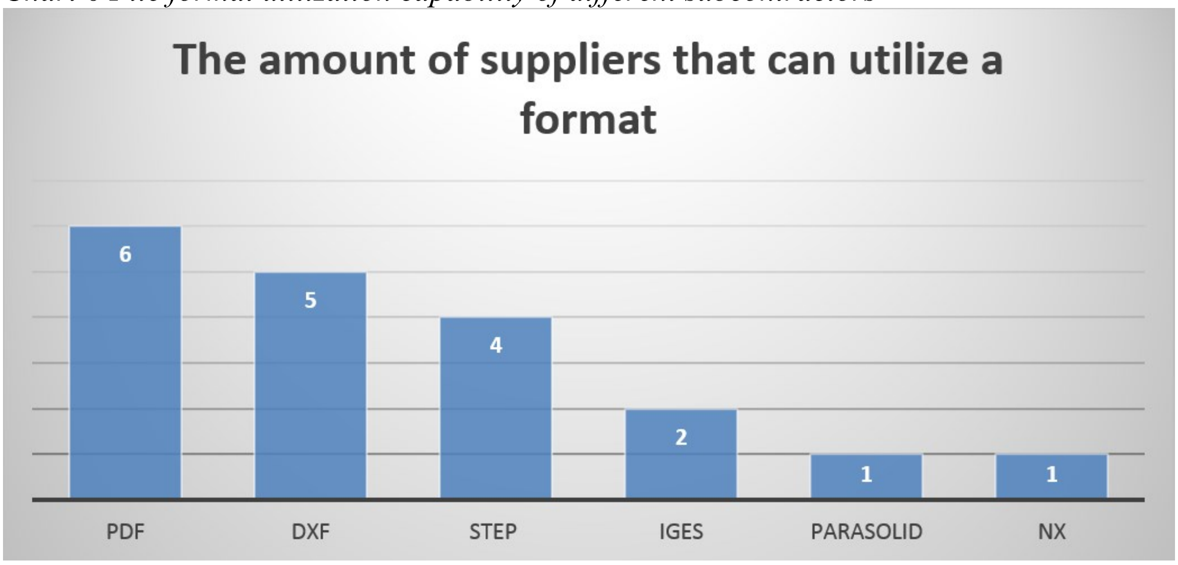
Supplier F can utilize STEP, DXF and DWG formats in their systems. The 3D-models can be utilized in the geometry checking, NC-programming and quality control. The quality control is typically done with basic methods for incoming materials, operations and the final inspection of the product. 3D-measuring is utilized for the initial samples of a batch and if required. The 2D-drawings are utilized generally in the production and quality control. In case there would be a 3D-model with the dimensions, the 2D-drawing should contain the missing manufacturing information such as manufacturing methods, tolerances and surface requirements. (Supplier F, 2015)

4.5.7 Conclusions of subcontractor survey

The subcontractor survey was conducted by determining the manufacturing processes of six subcontractors. The survey does not provide the whole truth about the situation among the subcontractors, but is a reasonable review about the present situation. The scale of the products manufactured by the suppliers was wide varying from thin sheet metal plates to castings and heavy steel structures.

Currently, Konecranes always delivers a 2D-PDF of the part to the subcontractor and usually a cutting DXF in case of sheet metal parts. 3D-models are also sometimes delivered, but that is not very common. Five suppliers out of six are able to utilize the 3D-models and the most typical formats are neutral formats STEP and IGES. Each supplier that is performing sheet metal cutting is capable of utilizing the cutting DXF in order to generate the cutting paths. Few suppliers also stated that even though the 3D-models could not be utilized straight in the manufacturing, it would be beneficial to have for example a 3D-PDF from which to check the geometry and possible lacking dimensions. One supplier admitted also that the capability of understanding the 2D-drawings is getting weaker among the professionals, because the 3D-models and 3D-visualizations are getting more popular. Chart 6 shows how the subcontractors can utilize certain formats.

Chart 6 File format utilization capability of different subcontractors



Many suppliers are capable of using both manual- and NC-controlled machines, but the manufacturing is almost only performed by the NC-machines. Each supplier is capable of utilizing CAM in their manufacturing processes. One supplier is not capable of utilizing 3D-models, but their manufacturing is mainly sheet metal manufacturing, in which the majority of the manufacturing is made in 2D. Four supplier out of six, are capable of utilizing 3D-models in their CAM-systems. These suppliers are performing tasks that benefit from the 3D-models including machining and casting. Some suppliers stated that sometimes they generate a 3D-model based on a 2D-drawing that is delivered from Konecranes. That 3D-model is later utilized in the creation of an NC-program. Similar task is also needed if a cutting DXF is not provided for sheet metal cutting. In conclusion, it can be said that a 3D-model and a cutting DXF should always be delivered straight from Konecranes to avoid extra work and possible errors in the model generation.

Even though the suppliers were generally well capable of utilizing 3D-models, they were not totally convinced that 2D-drawings could be taken out. The suppliers stated that all the same information that is delivered today should also be delivered in the future, even though the delivery format might be different. It was also mentioned that printed versions are still needed in the factory floor. Hence, it is beneficial, if that the annotated 3D-documents are easily printable.

By comparing the current situation among the Konecranes subcontractors to the U.S. Army subcontractors in 2009, it can be said that Konecranes subcontractors are more developed. The difference can mostly be explained by the six years difference in the survey year. However, Konecranes subcontractors are generally quite well capable of operating in the MBD-environment. The distribution of the Konecranes subcontractors to the same classes used within the U.S. Army subcontractors goes as follows:

- Class 2: 1 Supplier
- Class 3: 5 Suppliers

The requirements for the class 3 were that the supplier must be capable of somehow utilizing 3D-data and CAM in their manufacturing, and the manufacturing itself should be mostly performed by NC-controlled machines.

In the U.S. Army survey it was stated that the supplier has a minimum capability to operate in the MBD-environment if it is classified to the class 2 or higher. Hence, it can be said that all the six subcontractors are capable of operating in the MBD-environment. The biggest challenge is most probably going to be the compatibility of the different systems of Konecranes and the subcontractors. Hence, the neutral formats are going to be the most important data transfer formats.

4.6 Conclusions of Konecranes mechanical design & manufacturing

The majority of the mechanical design in Konecranes is made with the 3D-CAD NX. From the design point of view, the transition from current design approach to MBD, would not require major changes or investments. As a result of the design item surveys, the most typical manufacturing methods among Konecranes products are sheet metal manufacturing including sheet metal cutting and bending, as well as machining including turning and milling. Considering the sheet metal manufacturing, 3D-models are not commonly utilized, because the cutting is made based on a 2D-DXF. However, the 3D-model utilization among turning and milling is more appropriate.

The Konecranes manufacturing facilities in Hyvinkää are rather well capable of utilizing 3D-models. However, the majority of the programming within the rope drum and gear manufacturing is made with CAM-software that operate with 2D wire models. The first reason for that is that the products manufactured in the factories are often manufactured by turning, in which the 2D-CAM is an appropriate method. The models are typically also quite similar to each other and existing programs and macros can often be utilized in the program creation. The third reason is that SolidCAM as a program is not very familiar to the machinists, because there is no technical support or user trainings for it in Finland. However, the utilization of 3D-models and SolidCAM would offer certain benefits especially in the spare part manufacturing.

Similarly to own factories, the Konecranes subcontractors are also rather well capable of utilizing 3D-models and CAM-software. The manufacturing is done almost only by NC-controlled machines and the MBD capability class is 3 for almost all of the suppliers. However, the suppliers may require a printed version of the model for the manufacturing. Hence, it is beneficial if the annotated 3D-views of the model are printable. Due to the large amount of different CAD-software, the most important data formats in addition to the lightweight visualization formats, are going to be the neutral formats. Hence, it would be beneficial if the PMI-annotations in for example STEP-files would be usable by the various CAM-software.

In the beginning of the fourth chapter two basic Konecranes processes ETO and CTO were introduced. From the ETO point of view the MBD-approach would be very useful, because the ETO-projects contain a lot of design work, more varying products and more varying production chain. If MBD could decrease the time needed for the design, the projects would be finished more quickly. In addition, it would possibly help the manufacturers to produce their CAM-programs and thus parts faster. However, certain problems might occur, because the manufacturing chain is more varying meaning that the manufacturers might not be capable of utilizing the MBD-approach efficiently. In addition, the manufacturing processes and systems of the varying manufacturers may not be familiar to the designer.

From the CTO point of view, MBD might not offer as much benefits. The CTO-projects require less engineering and the manufacturing chain is relatively stable. In the process, only few changes are typically applied to the models or to the NC-programs. The most potential phases, in which the MBD benefits can be encountered are when the changes are applied. The changes are the points, in which the majority of the MBD benefits can be encountered. However, in case a certain change is made in a CTO-process, the MBD-approach would make the changing process most likely easier and faster.

5 MBD implementation to current environment

Chapter four determined the current practices among Konecranes mechanical design and manufacturing. The purpose of chapter five is to practically investigate how the MBD-approach would change the design process and how the models could actually be utilized in the manufacturing processes. Most of the data in chapter five is gathered from self-made observations, interviews and studies. However, certain written material and webinar publications have been utilized as a support.

The practical implementations from the design point of view contain two different sections. The first section includes investigations on how MBD could practically be implemented in the CAD-software. In addition, the behavior of the PMI-annotations among the visualization and neutral formats is investigated. The third chapter includes two time studies of how the design time is distributed into different tasks in the current design approach and in the MBD-approach.

The next two sections provide an overview of how the 3D-models are currently utilized in the manufacturing and quality control operations, and how MBD would give benefit to them. From the manufacturing point of view, three different methods are investigated. The first two sections include investigations of how the 3D-models are utilized in the CAM-program creation for the machining parts. In addition, MBD utilization in the sheet metal manufacturing and the 3D-model utilization in the creation of casting models are investigated. From the quality control point of view, one section gives an overview of how the 3D-models can be utilized in the coordinate measuring.

The last section is about how the MBD implementation should be executed according to user experiences. The section provides a step-by-step guide for the implementation process.

5.1 MBD implementation in CAD-software

This section considers how MBD can be implemented in CAD-software. It provides basic information on how the annotation work is performed and what kind of output files are available.

The CAD-software investigated in this section are NX and SolidWorks. NX is the main software utilized in Konecranes and SolidWorks is also utilized in certain operations. It is also beneficial to see how different software are handling the same issues Anark Core Platform is an add-in that can be utilized to provide a 3D-visualization material (e.g. 3D-PDF) from different CAD-software including NX and SolidWorks. The methods by which the software were studied, varied. The data for NX PMI was gathered by participating in an NX PMI training and by presentation held by the software vendor IDEAL PLM. The data for SolidWorks and Anark Core sections is mainly gathered from webinar presentations and the company web sites. In addition, a small study about different file formats was conducted.

5.1.1 NX PMI

Current CAD-software used at Konecranes is NX 8.5, which includes the PMI-functionality for adding 3D-annotations. The annotation tools include all the same functionalities that are available in the drafting environment, so a complete definition can be applied straight to the 3D-model. The PMI annotations are attached to certain model views, which define the visibility of the annotations. The visibility can also be defined by right clicking the annotation and selecting the views in which the annotation is visible. The annotation plane, which defines the orientation of the annotation, is also selected. The dimensions can be added to the model either by manually applying the dimensions between features or by displaying sketch or feature parameters as PMI. The other annotations, such as surface quality and geometric tolerances are applied manually. (Popkov, 2016)

To be able to deliver the product manufacturing information to the supplier, the model needs to be organized in a way that the information is easily available. From the specialized annotations menu, specialized product information (e.g. material information or BOM) can be applied to the model. In addition, section views and lightweight section views can be created in the PMI-environment. The section views can also be annotated similarly to the other views. The PMI can also be extracted from the model to the drawing, which makes the drawing creation very fast in case a drawing is needed. (Popkov, 2016) NX uses a JT-file as a visualization format. Figure 28 presents an annotated JT-file opened by the software JT2Go.

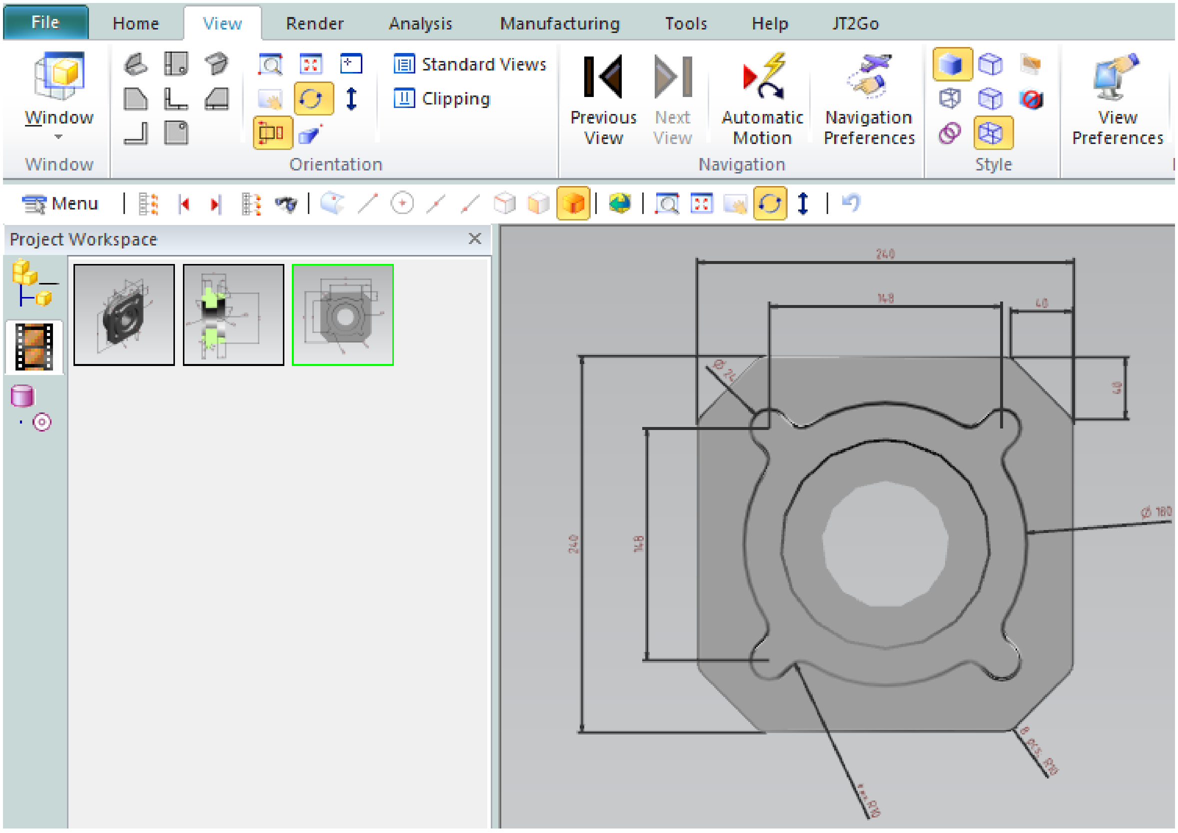


Figure 28 Annotated JT-file in JT2Go

The PMI-annotations of NX can be reused in several applications by Siemens. NX CAM can utilize the PMI annotations while generating the machining paths. The surface quality and tolerance annotations define for example the machining parameters and offsets to the nominal dimension. Certain rules need to be predefined before the program creation. One of the rules is to define the point of the tolerance margin in which the machine tool is targeting. (Simons, 2016)

Another reuse possibility is a tolerance analysis performed by Teamcenter visualization. The tolerance analysis can be done based on an annotated JT-model. The analysis applies different values for the dimensions within the tolerance margin and performs a statistical analysis for the given values. Thus for example a compatibility of the assembly parts can be checked. (Simons, 2016)

The NX PMI annotations can also be utilized in CMM Inspection software in order to create the measuring program. The program is created almost similarly to the CAM-program and the PMI dimensions can be utilized in the creation of different measuring routines (e.g. quantity of the measuring points) based on the tolerances. (Simons, 2016)

5.1.2 SolidWorks MBD

SolidWorks MBD is an add-in available in SolidWorks 2015. The purpose of the add-in is to help in defining, organizing and publishing of 3D-PMI. The dimensioning of the part can be made manually or semi-automatically by utilizing SolidWorks dimXpert add-in. The additional annotations such as surface quality and welding annotations are added manually from the SolidWorks MBD CommandManager. 3D-reports, that are designed to replace the 2D-drawings, can be published as a 3D-PDF or as an eDrawings, which is SolidWorks' own format. (SolidWorks, 2015)

An important section in addition to the annotation work, is the capturing of 3D-views. With the command "Capture 3D view", user can save selected views, such as front and top, with relevant annotations. A typical way is also to include an isometric view with all the dimensions. To improve the visualization, section views and exploded views can also be generated. The captured 3D-views can later be included in the 3D-report. (SolidWorks, 2015)

In a webinar organized by PLM Group, the process of generating a 3D-report with PMI, was introduced by a solution manager Ola Claesson. The webinar was basically a step-by-step guide on how to add the PMI to a SolidWorks' 3D-model. The webinar also introduced the 3D-PDF and eDrawings documents created by SolidWorks MBD. The view generation of the 3D-views began so that the model was first oriented to the desired orientation. After that, all the annotations that were supposed to be visible in that view, were selected. Finally, the annotated 3D-model was be published as a 3D-PDF. The 3D-PDF included all the captured views and other product information. The model could also be measured from the 3D-PDF in case a dimension was missing. In addition, the templates of the 3D-reports can be edited depending on the requirements. (Claesson, 2015) Figure 29 presents a 3D-PDF generated by SolidWorks.

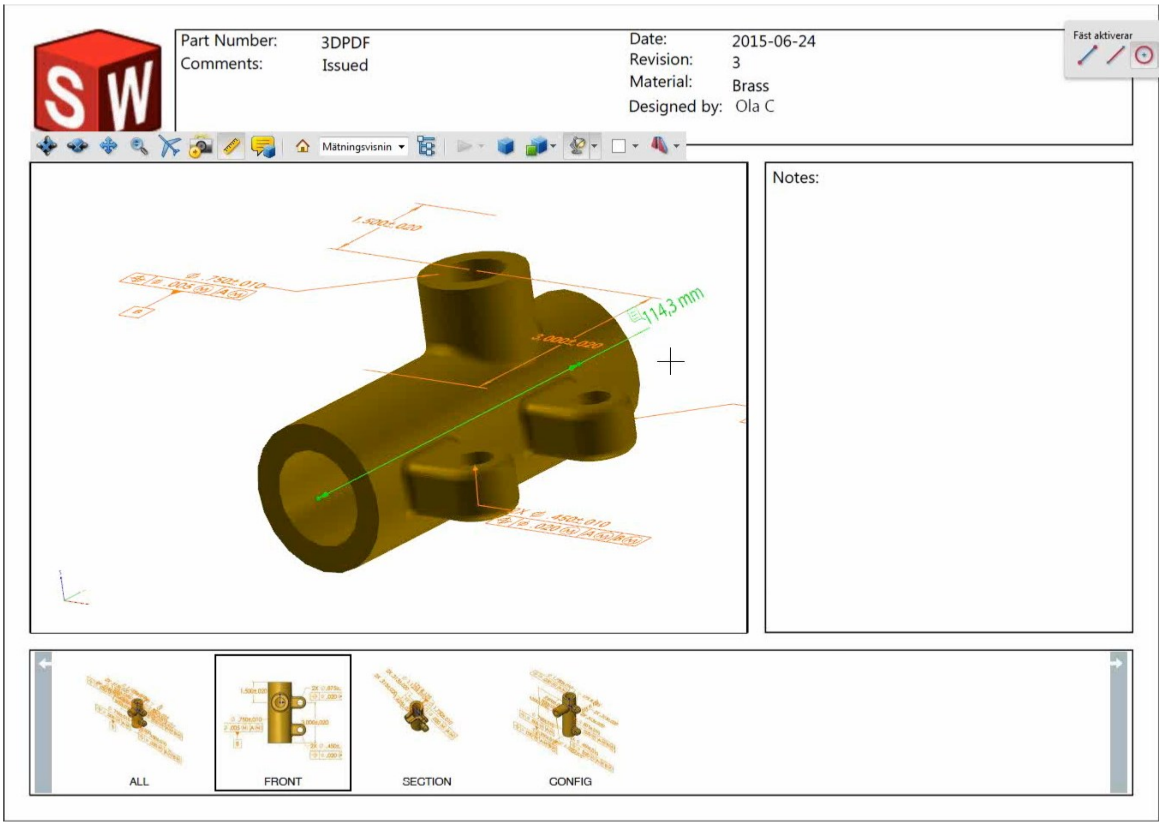


Figure 29 3D-PDF of a part including several views (Claesson, 2015)

5.1.3 Anark Core Platform

Anark Core platform is a software platform, which can be utilized in order to transform native CAD-data to a 3D-PDF or a 3D-HTML dataset. The process can be made automatic, meaning that when the original 3D-model is changed, the changes apply to the 3D-PDF or 3D-HTML. The Anark Core supports several input data formats including native formats from NX, SolidWorks, Creo and Catia and neutral formats including Parasolid, PDF and JT. (Anark, 2016) The Anark Core is utilized by several large companies around the world including NASA, Boeing and Ericsson. (Rosenqvist, 2015)

The Anark Core contains two basic components. Anark Core Workstation is a tool that can be utilized in order to create digital work instructions and “recipes”, which are used to automate the document creation process. It can also be used to make templates for 3D-TDPs. Anark Core Server is the center of all the functions. It is utilized to automate the creation process of the TDPs. The Server can use the recipes created by the Workstation. Both the Anark Core Workstation and the Server have integrations to the CAD-software SolidWorks, NX, Catia and Creo. The Anark Core has also an integration to the PLM softwares Teamcenter, Windchill PDMLink, Enovia and SolidWorks EPDM. (Rosenqvist, 2015)

The Webinar presented by Rosenqvist (2015) introduced a 3D-TDP created by the Anark Core. Figure 30 presents a 3D-PDF document created from SolidWorks model. The 3D-PDF had an embedded STEP-model in it. The example document was made for an assembly of a motor casing. It included all the required information of the assembly such as BOM, dynamic views with envelope dimensions and dynamic exploded view with balloons. All the individual parts could be selected from the document and investigated further if required.

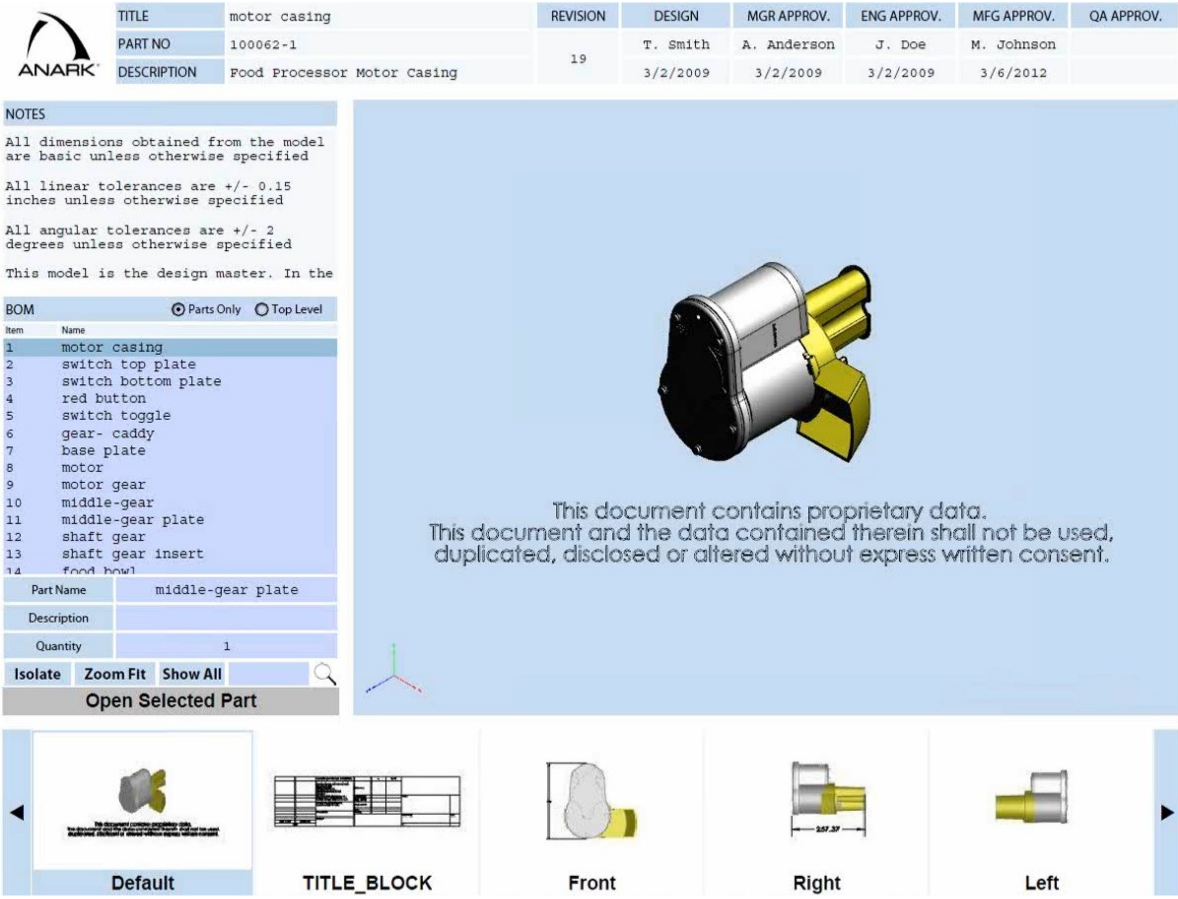


Figure 30 A 3D-PDF of an assembly created by Anark Core (Rosenqvist, 2015)

5.1.4 Evaluation of MBD related file formats created by NX

As mentioned in Chapter 3, there are several file formats related to MBD. In addition, systems and software utilized among different organizations vary so much that the neutral formats are going to be important in the collaboration. Hence, it is beneficial to practically see how the PMI-annotations can be transferred from a native CAD-file to various different formats.

To determine how the different formats behave, a small study was performed. An annotated NX-file was exported to different formats, and the exported formats were opened with NX, SolidWorks and Creo. The different formats investigated were STEP 203, STEP 214, IGES, Parasolid, JT and 3D-PDF. The 3D-PDF was exported from Teamcenter, because NX cannot export it. The 3D-PDF file was investigated by opening it with Adobe Reader and the JT-file was opened with viewers JT2Go and XpresReview. The inspection process for the rest of the formats can be seen in Figure 31.

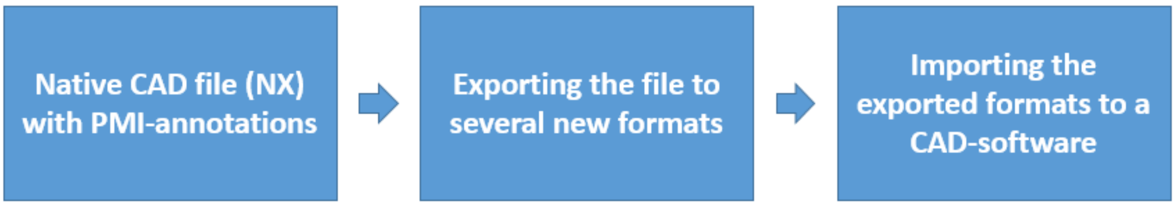


Figure 31 The process of evaluating the file formats

The both STEP formats 203 and 214 behaved basically similarly. The only difference was determined in the colors of the models. The PMI-dimensions could be included to both of the formats by selecting them in the options in both exporting and importing phase. The annotations followed to each basic view similarly as they were created in NX. However, a lightweight section view created in NX PMI did not follow to the STEP-model. A basic section view generated in NX followed to the STEP-file as an individual body. In addition to the basic views, the STEP-files automatically created an additional "global PMI view" in which all PMI-annotations were applied. Even though the PMI followed quite well to the STEP-files, the link to the features was not working after the export. When selecting an annotation, no highlighting of the features was applied. In addition, the PMI viewing direction did not update when rotating a model, and the annotations could not be selectively hidden or unhidden.

One of the file formats investigated was IGES. While exporting an IGES-file, there was no selection possibility whether to include the PMI or not. However, the PMI-annotations followed to the IGES-model, but the annotation views followed to the model in an illogical way. The annotations remained in certain views, but disappeared in others. In addition, the section views did not follow to the model correctly even though they appeared in the model tree. In the Parasolid-file, no PMI was managed to get exported.

An important thing within the study was to determine how a JT-file behaves in the free viewers, and whether it is possible to open it with different CAD-software. The both viewers JT2Go and XpresReview were able to open the file and usage of the viewers for the visualization was simple. All the PMI-annotations and all the basic model views as well as the lightweight section view followed from NX to the JT-model. In addition, the PMI could be selectively hidden or unhidden. All the associated features also got highlighted when an annotation was selected, and the viewing direction updated when the model was rotated. The both viewers also included the possibility to take measurements. In addition, JT2Go included the possibility to make additional section views and take "snapshots" that can easily be printed. All the model views that include PMI, were automatically added as snapshots. CAE-results could also be seen from the JT2Go, if they were included in the model.

In addition to JT, 3D-PDF can be utilized as a visualization format. 3D-PDF could not be exported from NX, but the exporting via Teamcenter was possible. However, the file exported from Teamcenter could only be utilized to show the geometry, because PMI-annotations were not applied to it. Table 2 shows how the different features followed from the NX-file to the neutral file.

Table 2 Possibility to utilize the features of an NX-model in other formats

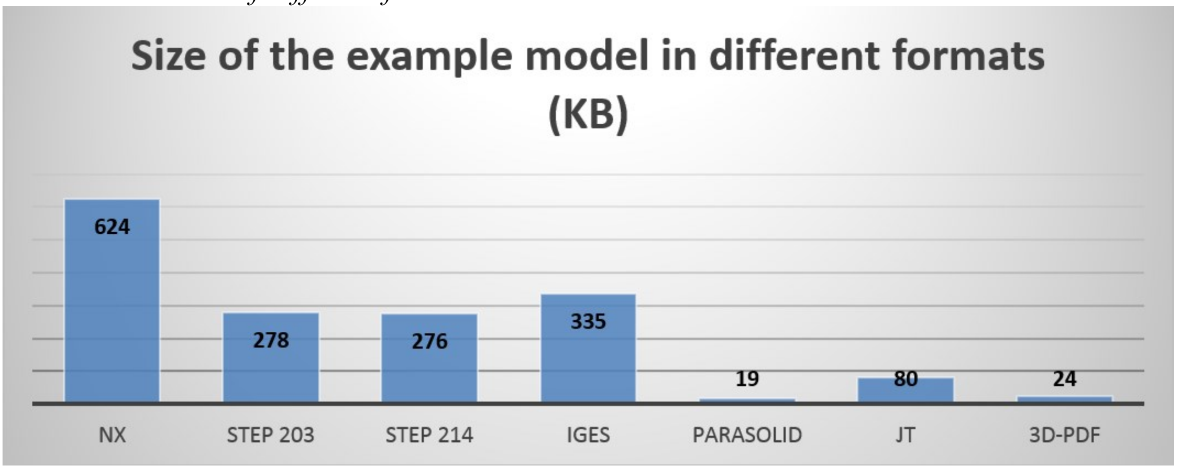
Feature	STEP 203	STEP 214	IGES	Parasolid	JT	3D-PDF
Geometry	x	x	x	x	X	x
PMI	x	x	x		X	
Basic model views	x	x		x	X	
Lightweight section view					X	
Section view	x	x				
PMI associated to features					x	

One part of the study was to test how the different formats created by NX, open with SolidWorks2014 and Creo 3.0. The both software opened STEP, IGES and Parasolid, but

JT could not be opened. The PMI could be shown with Creo for both of the STEP-files, but not for IGES. The annotations were not associated to any feature similarly as they were in NX. With SolidWorks, no PMI was managed to get opened.

One issue that might be interesting, is the difference of the file sizes. Both of the STEP-files and IGES are assumed to be “heavier” files that can be used in several applications, while JT and 3D-PDF are assumed to be lightweight visualization formats. File sizes of the exported model were taken in different formats. Chart 7 shows that the original NX file is the largest and STEP and IGES come after that. An interesting issue is that the Parasolid-file is the smallest of them all.

Chart 7 File sizes of different formats



In conclusion of the formats STEP and IGES, it can be said that STEP works generally better. The both formats include the PMI-annotations, but in the STEP-files they are organized in the same views as in the original model and no annotations are disappeared during the export. However, the association between a PMI-annotation and a feature disappears in the export. In addition, the annotations are no longer visible in the history tree and cannot be hidden or unhidden. In addition, the viewing direction does not update according to the orientation of the part.

From the visualization format point of view, JT suits better to the current environment. It can be viewed with free viewers and the visualization follows the standard ISO 16792. JT would be an excellent format if it could be utilized in other operations such as CAM. In the current environment, 3D-PDF can only be utilized for checking the geometry of a part or an assembly. However, 3D-PDF would be a suitable option, if the PMI-annotations could be applied to it.

5.2 Time studies

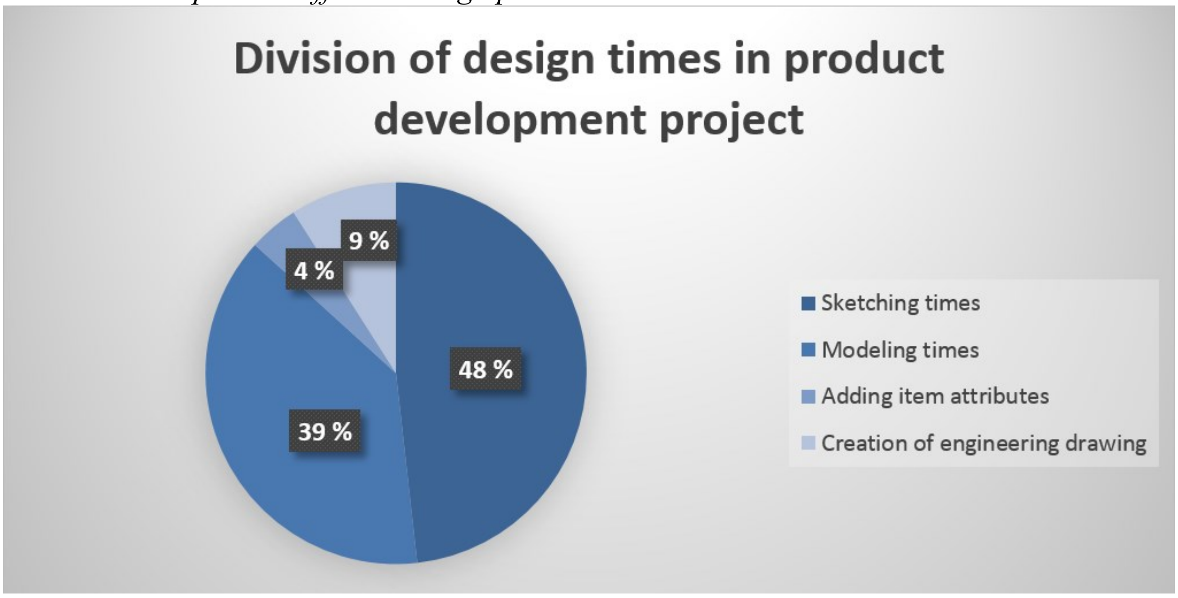
During autumn 2015, a small study of the time spent in different tasks of creating a design item, was conducted. The items were created for a product development project and different tasks measured were sketching, modeling, adding item attributes and creation of an engineering drawing. There were two mechanical designers involved in the study. The feedback from the designers was that the design process is iterative and it is difficult to separate the sketching and modeling phases. However, in this study the most important result was to determine the percentage of the creation of the 2D-drawing in relation to the time spent in the entire process.

The parts that were designed in the product development project were mainly sheet metal parts and they were dimensioned only to a traditional 2D-drawing. The results achieved from the design item survey (UM trolley and Q-hoist) prove that the machining parts are also common. In addition, one objective of the thesis was to compare the time spent in dimensioning the parts by traditional method and by PMI-dimensioning. Hence, another study, which measured the time spent in the modeling and dimensioning phases was conducted. The parts involved in the second study were mostly manufactured by machining, and they were dimensioned by both traditional method and PMI-dimensioning.

5.2.1 Time study results of product development project

The parts designed in the project were mainly laser cut and bent sheet metal parts. In addition, there were other parts such as a welding assembly and a cover made by 3D-printing. Chart 8 presents the distribution of time spent in different design phases.

Chart 8 Time spent in different design phases



In many occasions, it has been estimated that the drawing creation takes around one third of the design time. In this case, it took only 9 %, which can be explained with a couple of reasons. The first reason is that the parts were generally laser cut sheet metal parts, which are manufactured based on DXF-models. Hence, certain simplifications could be made to the drawings. Another reason is that sheet metal parts usually do not contain a significant amount of tolerances, which typically increase the drawing creation time. In addition to the sheet metal parts, the project included also a cover that was made by 3D-printing. In that case, only a very simple 2D-drawing with boundary dimensions was generated and the manufacturing was made based on only the 3D-model. The design of the part was also relatively complex and the sketching and modeling of it took quite a large amount of time. Excluding the time spent with that part, the percentage of the drawing creation rises up to 12 %. If the project would have included mainly machining parts with a lot of tolerances, the percentage would have probably been higher.

5.2.2 Time study results of PMI-dimensioning

The study was conducted during the January and February of 2016. Because there was no suitable project proceeding, in which the study could have been included to, it was made by modeling and dimensioning already existing parts. The first step of the process was to make a new 3D-model by utilizing an existing 2D-drawing of the part. After the model was created, the next step was to apply the dimensions and other annotations to the model and create an additional drawing in which the annotations were also applied. NX PMI (NX 8.5.) was utilized for the creation of the PMI-annotations. The process can be seen in Figure 32.

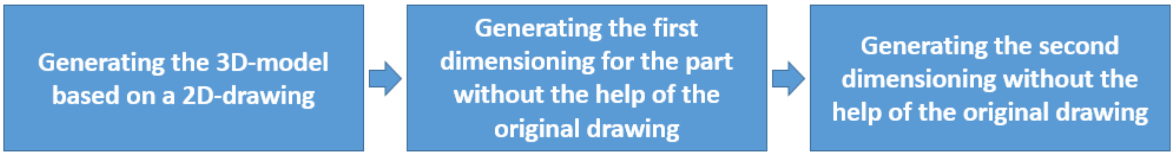


Figure 32 The process of the second time study

The study contained four items and three designers meaning that each 3D-model, 2D-drawing and PMI-dimensioning was generated three times. The same annotations were applied to the model and the drawing and the time spent in modeling, PMI-dimensioning and drawing creation were measured. The items involved in the study are listed below:

- Shaft
- Wheel
- Bearing housing
- Rope clamp

Chart 9 Time spent in different tasks

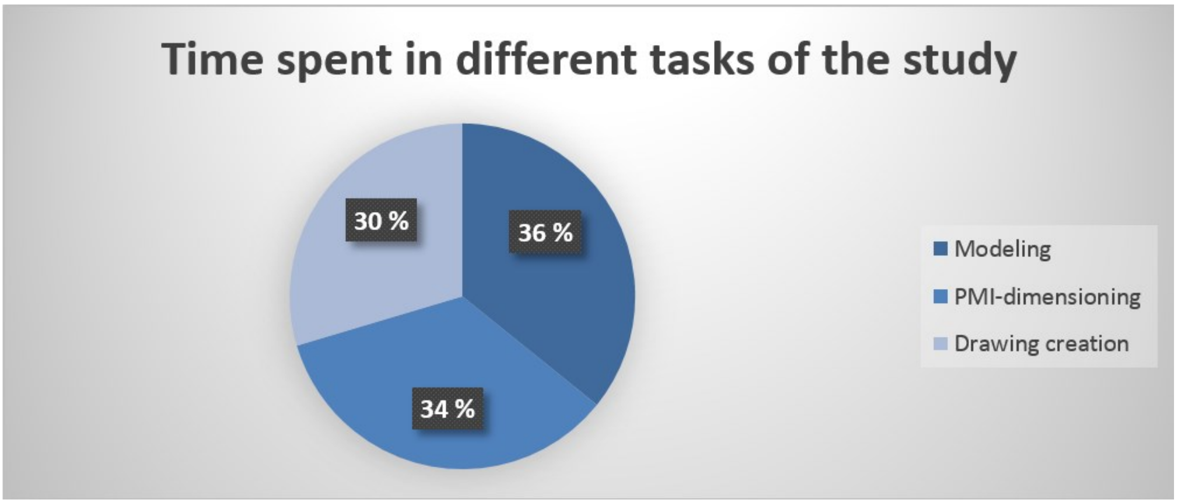


Chart 9 shows the time spent in the different design phases. As the results show, the PMI-dimensioning took actually a bit longer than the dimensioning of the drawing. There are several reasons that might explain the result. The first reason is that the designers are not yet fully familiar with the PMI-tools and the drawing tools are very familiar instead. Another reason is that some of the PMI-tools were not working as expected (e.g. hole dimensioning was difficult), which caused certain confusion.

In this time study, the dimensioning took about 83 % compared to the time spent in the modeling. The percentage is quite high compared to the PD project in which the dimensioning took only about 25 % compared to the time spent in modeling. The difference can be explained by the difference of the models created within the studies. In the second study, the models were created for machining items and for one casting. All the dimensions and annotations, including necessary tolerances and surface qualities were applied, while in the PD project only the main dimensions were applied.

5.2.3 Conclusions of the time studies

In conclusion of the first study, it can be said that the time spent in the drawing creation of a sheet metal parts is relatively small, about 10 %. However, the dimensions and other information that were added to the drawing were basically all included to the 3D-model and the DXF-model. Hence, the drawing creation was basically unnecessary work. That kind of dimensioning could also possibly be excluded in the machining parts supposing that the dimensions could be manually checked or automatically gathered from the 3D-model. By adding the tolerances and other annotations to the model, the drawing creation could be excluded in the machining parts as well. This would increase the modeling time, but the elimination of the drawing creation could still save about the same amount of time that could be saved with the sheet metal parts, meaning 10 %. The creation of the 3D-printed plastic cover is a good example of how the design and manufacturing process would proceed in the model-based environment; the manufacturing is made totally based on the 3D-model and the 2D-drawing is not necessary.

In conclusion of the second study it can be said that the PMI-dimensioning does not currently save time in the design phase. However, familiarizing with the PMI-tools and applying the sketch and feature dimensions of the model straight to PMI-dimensions might make the PMI-dimensioning as fast as or even faster than the drawing creation. In addition, the current time difference between the PMI-dimensioning and the drawing creation is quite small. Hence, the PMI-dimensioning would be a reasonable way to do the dimensioning due to the other benefits it offers.

5.3 3D-model utilization in manufacturing

This chapter investigates how the 3D-models can be utilized within different manufacturing processes. Based on the results of the design item study, the most typical manufacturing methods are sheet metal manufacturing and machining, which are both investigated in this chapter. In addition, the creation of casting models based on 3D-models is also investigated, because an opportunity to see the process emerged. Most of the observations in this chapter are based on own observations and interviews. An exception is the next section, which is a summary created based on a presentation.

5.3.1 3D-model utilization in Computer Aided Manufacturing (CAM)

K. Kuutela & P. Manninen from Pathtrace Oy gave a presentation on 27.11.2015 about 3D-model utilization in CAM. Pathtrace Oy is a Finnish company that is specialized in NC-programming, post-processor-programming and NC-simulation. The software they utilize in the NC-programming is EdgeCAM and the software utilized in the NC-simulation is CGTech Vericut. The presentation gave an overview on how the 3D-models are currently utilized in CAM and which benefits and problems are encountered in the use. The

manufacturing methods discussed in the presentation were turning and milling. (Kuutela & Manninen, 2015)
The most typical file formats that are currently utilized among the Pathtrace software are STEP and Parasolid. The first step in the generation of an NC-program with CAM is to create the billet model from a 3D-model of the part to be machined. The billet model and the original model are then transferred to a selected machine tool within the CAM-software and the needed tools are selected from the tool library. After that, the machining phases and machining paths are generated from the billet model and the 3D-model of the part. The next step is the simulation, which provides the results of machining time etc. The final step is to post-process the NC-program to make it usable for a certain machine. (Kuutela & Manninen, 2015)

A major part of the presentation was to clarify how the 3D-models can be designed to make them easily usable for the CAM-software. According to Kuutela & Manninen, the 3D-models are currently not completely fulfilling the requirements that are set by the CAM-programming. Hence, it is typical that the CAM-programmer needs to edit to the model and sometimes recreate the model from scratch. If the original 3D-model is edited, certain problems may occur and duplicated work may need to be done. If the CAM-model and the original CAD-model are separated, the possible updates and modifications must be applied to both of the models instead of one. Another possible issue is that the revision control does not work and the CAM-model and NC-program in use do not correspond to the updated CAD-model. (Kuutela & Manninen, 2015)

There are several examples of problems that are encountered during the creation of NC-programs generated based on 3D-models. Based on the problems, several development ideas were given. The list of the most important development ideas is given below:

- Open discussion between the CAD-designer and the CAM-programmer
- Dimensioning to the middle of tolerance range (in case the CAM does not read the PMI-annotations)
- Emphasizing the faces that should be machined
- Thread information to the 3D-model
- Surface quality information to the 3D-model
- Information of the available tools to the designer (as an example roundings can be made to have a correct radius)

5.3.2 Prototype manufacturing at PMC

During the creation of the thesis, the mechanical design department received a task to design clamps for fixing two I-beams together. The manufacturing was decided to be performed at the PMC. The clamps were going to be manufactured using an NC-controlled milling machine and the NC-programming was going to be made with SolidCAM utilizing the 3D-model of the clamp. Hence, it was a good opportunity to evaluate the process, in which the parts are manufactured without a 2D-drawing. The clamps were manufactured from a 60 mm aluminum plate and the machining was performed with a milling machine and a rotary table by HAAS. Figure 33 shows the 3D-model of the part.

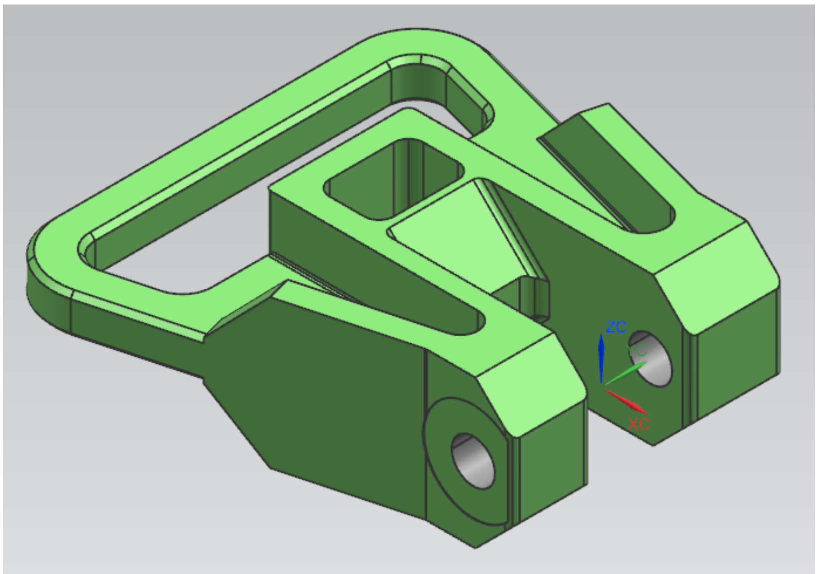


Figure 33 3D-model of the part

The manufacturing process began by generating the NC-program. The first step of the process was to open an IGES-model of the clamp by SolidWorks. After that, the SolidCAM 2010 application was opened and the correct manufacturing method, in this case milling, was selected. The next steps of the program creation were to define the machine to be used and generate the billet model. A default billet can be automatically generated by SolidCAM. After that, the billet model was applied to the CAM-program. The next step was to apply several machining phases with different tools to the billet to achieve the geometry of the 3D-model of the clamp. In this case, the machining is first performed from the upper side and after the side is finished, the part is turned around and machined with the same operations from the lower side. Each machining phase can be simulated to see how the machine is actually performing the machining and to determine, if any collisions or other issues occur.

The creation of the NC-program took about four hours. Currently, machinists are not fully familiar with SolidCAM, because there is no technical support for the software in Finland. There were also a couple of problems with the model and slight changes needed to be applied to it. The machining of one clamp took totally about three hours and it was conducted in two phases as mentioned in the previous paragraph. The machining of the first phase took about two hours and the machining of the second phase about one hour. Another NC-program was also created for the first machining phase with newer SolidCAM version 2014. No machining has yet been performed using that program, but the simulation of the first phase resulted in a machining time of one hour. It can be estimated that the machining of the whole part would take about 1.5 hours with the new NC-program.

In conclusion, it can be said that the machining of prototypes is possible without a 2D-drawing. The clamp did not contain any tolerances or specific surface quality requirements, meaning that no PMI-annotations were needed either. In case the tolerances or other requirements would exist, the PMI-annotations would be useful. The benefits of the PMI-annotations could be fully utilized, if the CAM-software could automatically extract them. However, they would be useful also if they were visible in the model and the machinist could manually apply them to the program. To make the process more efficient, advanced training on SolidCAM should be given to the machinists. It would speed up the creation and increase the quality of the NC-programs. Another important issue is the co-operation of the CAD-designer and the CAM-programmer. By improving the co-operation, both parties would more likely understand the key features needed in the processes. It would also be useful for the designer to know the existing tools, so for example rounding radiuses could be selected depending on the tools available. The observations made during the NC-program creation of the clamp, correspond well to the development ideas given by Kuutela & Manninen.

5.3.3 Sheet metal manufacturing

An important manufacturing method among Konecranes products is the sheet metal manufacturing including cutting and bending. To gather a proper knowledge about the process, a visit to a subcontractor that performs laser cutting and bending, was organized. The subcontractor has two laser cutters and four edging presses. The maximum thickness for the cutting is 15 mm of black iron and the maximum bending thickness for black iron is 8 mm. The supplier has a 3D-CAD software Geomagig design, a CAM-software Bystronic Bysoft for the laser cutters and a simulation software Amada BendCAM for the bending presses.

There are basically three different file formats that are utilized at the subcontractor: 2D-PDF, DXF and STEP. 2D-PDF is the basic file that is always required from a customer. It is utilized as an attachment to their part identification system and as a printed version in the production line. Their opinion was that a printed version is required to be able to perform the production. A cutting DXF is utilized in the creation of the cutting paths. It is also required from the customer, because otherwise the subcontractor must create it them self. This would enable errors and increase costs. The STEP-files are utilized to check the compatibility of assemblies and to check possible lacking dimension that are not represented in the 2D-drawing. A flat pattern view of a bended sheet metal part can also be generated by the STEP-model, if the model has been created correctly.

The basic process of the sheet metal manufacturing at this subcontractor goes as follows. The customer delivers a 2D-PDF and a cutting DXF and the supplier creates an item ID that embeds the 2D-drawing. The next step is to create the laser cutting path based on the DXF-model. The cutting path can be generated automatically and quickly, if the DXF is made correctly. After that, the parts to be cut from the sheet are nested by a nesting software. The nesting can be made either automatically or manually. The next step is to program the edging press with the help of the press simulation software. After that the part is cut by a laser cutter and bent by a press. Figure 34 presents the sheet metal manufacturing process.

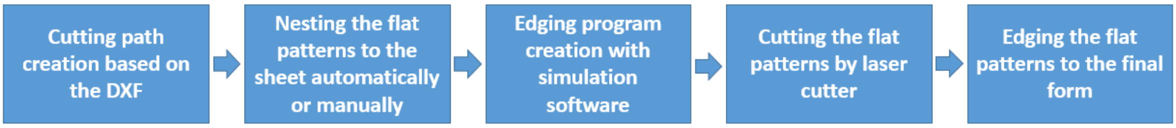


Figure 34 Manufacturing process of bended sheet metal part

5.3.4 Creation of casting models

A significant manufacturing method is casting, and one of the casting methods is sand molding. An important part of the sand molding is the creation of a casting model. The casting model is utilized in order to create a casting mold and it needs to have the same geometry as the part to be casted. During February 2016, a visit to a company that creates casting models was made. The company creates most of the models to a foundry nearby, but some of the models are also supplied to other foundries.

The creation of the casting model requires a 3D-model, which is typically delivered by the customer. The 3D-model can also be made based on a 2D-drawing, but naturally the better way is that the 3D-model is delivered by the customer. The 3D-model is utilized to create an NC-program with CreoCAM and a 2D-drawing is not needed in the process. The casting model is later created using this NC-program. The models are typically manufactured by an NC milling machine and the most typical material for the casting models is polyurethane plate. As a feedback, the company representatives suggested that the customer should deliver both casting model and a machining model, so they can confirm that there is enough machining allowance. Figure 35 presents the creation process of a casting model.

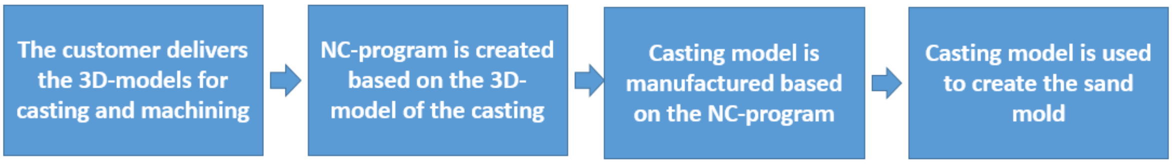


Figure 35 Creation process of a casting model

5.3.5 Conclusions of the manufacturing processes

The conclusions of the 3D-model utilization in turning and milling are based on the presentation by Pathtrace and by the own observations during the prototype manufacturing at PMC. The most important thing observed is that machining is possible to be performed only by a 3D-model. However, the process is not always simple and the model may need to be edited. Hence, the co-operation between a designer and a CAM-programmer is essential. No PMI-annotations were utilized or needed in the prototype manufacturing process. However, it would be useful to have them in case the part to be manufactured contains a lot of tolerances or surface quality requirements. In the best case, the CAM-software would automatically read the PMI-annotations and generate the NC-program taking them into consideration. The PMI-annotations would also help the CAM-programmer by only being visible. Surface qualities, tolerances etc. could then be manually applied to the NC-program without the help of a 2D-drawing.

From sheet metal manufacturing point of view, the 3D-models are not as beneficial. The largest benefits can be encountered in the visualization of complex geometries and in the checking of the compatibility of large assemblies. For that purpose, the 3D visualization formats (JT & 3D-PDF) are well suitable. The sheet metal cutting is already basically utilizing MBD-approach. The cutting paths are automatically generated from a cutting DXF and no 2D-drawing is needed. The DXF-file is commonly utilized among the sheet metal manufacturing, so it cannot be supposed that any other format (e.g. STEP) would replace it in the near future.

The creation of a casting model can also be performed based on only a 3D-model. The model is utilized in a CAM-software to create an NC-program, and a 2D-drawing is not needed or utilized. However, it is beneficial to deliver 3D-models for both casting and machining, so the casting model creators can ensure that there is enough allowance for the machining. Because the creation of a casting model does not require a 2D-drawing (or 3D-annotations), it can be said that the 3D-model does not require the PMI-annotations either. However, the machining model should include the PMI, especially if there are any tolerances or surface quality requirements.

5.4 3D-model utilization in coordinate- & camera measurements

The coordinate measuring machine (CMM) demonstration was given by a Finnish company that is specialized in 3D-measuring and high speed cameras. The CMM utilized in the demo was Nikon VMA – 2520, which has both touch probe and vision measurements. The touch probe measurements are very reliable and they work better especially for spherical and glossy specimen, while the vision measurements are significantly faster. The vision measurements are also usable in certain special cases where the touch probe cannot be used. The special cases include the measuring of a small detail, measuring of a marking on a surface or measuring of a detail behind a glass.

The CMM is connected to a software CMM-manager, which is utilized for creating the measuring programs and conducting the measurements. Typically, the measurements require two elements, the actual part to be measured and a 3D-model of the part. A slightly simplified process of the coordinate measurements can be seen in Figure 36. The first step of the process is to set the origin to the desired location in the 3D-model. The location can be for example a crossing point of two edge lines. The next step is to merge the coordinates of the model and the actual part. The coordinate merging could be avoided with a jig that would locate the part automatically in a predefined location.

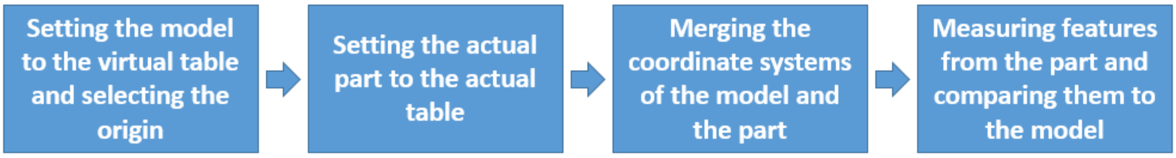


Figure 36 Simplified process of coordinate measuring

There were several different measurements for the touch probe that were demonstrated during the visit. The measurement paths were created automatically after the user had selected the features to be dimensioned. The measurements made during the visit were all done for a chain guide and several results from the measurements were achieved. Figure 37 presents the measurement report generated based on cylinder measurements of four holes of the chain guide. The report provides the nominal and actual values of the coordinates, diameters and distances and also the deviation between the nominal and the actual value. Figure 38 shows how the measurement of a curve is performed. The measurement points are visible in the figure and the deviation to the nominal coordinate is presented by colors. The green dot means that there is very small or no deviation, the red dot means that the coordinate is at least 0.1 mm smaller than the nominal and the blue dot means that the coordinate is at least 0.1 larger than the nominal.

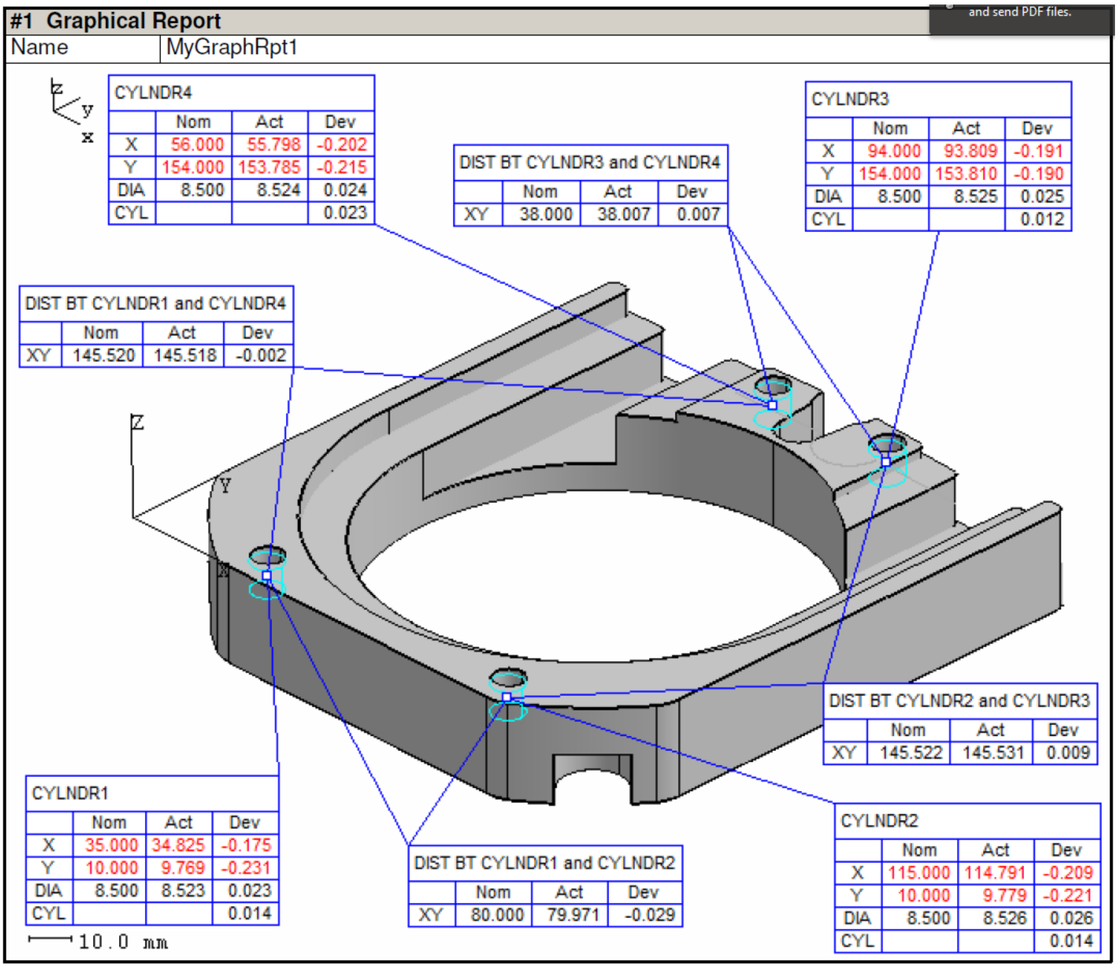


Figure 37 Results of probe measurements of cylinders

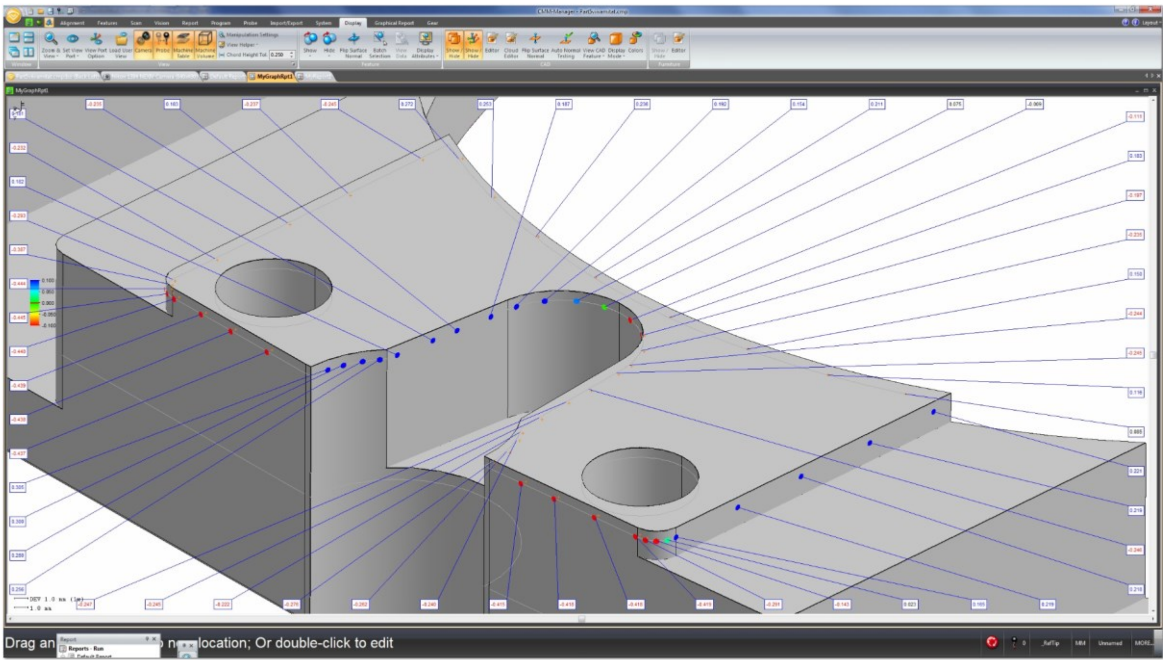


Figure 38 Results of curve measurements

In conclusion of the coordinate measuring, it can be said that the 3D-model is basically essential for performing the process easily. The most time consuming phase is the merging of the coordinate systems, which could be avoided if a jig would be utilized. The program creation itself is rather simple. The features to be measured are selected from the 3D-model and CMM-software creates the measuring paths automatically. The PMI-annotations could be utilized in the process by automatically notifying if certain dimension is out of the tolerance range. Another application for the PMI would be the creation of a CMM-program, in which the PMI could be utilized for example to add more measuring points to a spot where the tolerances are tight. However, the PMI-annotations could not be utilized in the current version of the software CMM-manager.

5.5 How-to guide for adopting PMI

A report "*The How-to Guide for Implementing PMI*" from a company Tech-Clarity created by M. Boucher (2015) includes customer feedback about the implementation of the MBD-environment utilizing the NX PMI. In addition to the feedback, it provides a step-by-step of implementing an MBD-environment. The most typical benefits given by the companies that have implemented the PMI-annotations in the 3D-models are listed below:

- Less time spent in the creation of the dimensioning & tolerancing
- Less misunderstandings
- Less errors and duplicated work, because the information exists in a single location

However, there have also been certain issues encountered during the implementation process. The most typical issues are listed below:

- All the changes cannot be made immediately. A step-by-step approach should be used and each step should be well planned
- Resistance for the cultural change
- Difficulties to change the working methods of each party involved

The step-by-step guide generated by Boucher (2015) is presented below:

1. Create an extensive plan on how to implement the PMI-annotations. It is likely that resistance for the cultural change will exist
2. Define the departments and the persons inside the company, who should be involved in the project
3. Create and maintain a working culture that supports the change
4. Execute a training for those who are going to use the PMI-tools
5. Introduce the PMI-files gradually to the subcontractors so they will learn the value of the approach themselves
6. Remember that the implementation of the PMI does not have to be on/off. It can be executed by gradually moving away from the 2D-drawings
7. Start with a pilot project that is small enough to be kept in control. Too large changes at once will not work
8. Learn the best practices from the pilot project(s) and execute them

5.6 Conclusions of MBD implementation

The implementation of the MBD-environment requires several changes to the processes. From the designer's point of view, the major changes would be seen in the CAD-modeling. The current software used in Konecranes is NX 8.5. It includes the PMI-application, in which the possibility to implement the MBD-environment exists with no additional investments.

NX creates a JT-file always when saving a model, and the JT can be used for the visualization of the part. NX PMI also maintains the possibility to create 2D-drawings from the annotated 3D-views of the model. SolidWorks MBD, which is available in SolidWorks 2015, has a slightly different approach for the document sharing. In that application, the user can create a 3D-PDF, which is basically a dynamic drawing. The dynamic drawings look like traditional drawings, but they have annotated 3D-views that can be zoomed and rotated. By using Anark Core platform, 3D-PDFs could also be created from the NX-models almost similarly they are created in SolidWorks. However, the using of Anark Core might not be reasonable, because it would require additional investments.

As a part of this master's thesis, two time studies were conducted. The purpose of the study regarding to the current process was to achieve an overview of how long time each design phase takes. Several conclusions were achieved from the study. The creation of the 2D-drawings for the sheet metal parts require a lot less time compared to machining and casting parts. The reason for that is most likely that the 2D-drawings of sheet metal parts were not fully annotated, because the cutting geometry is achieved from a cutting DXF. Generally, the machining and casting parts have been fully annotated and especially machining parts can contain a lot of tolerances and surface quality requirements. On the other hand, the 3D-models for casting would not need to be annotated at all, because only the 3D-model is utilized in the CAM-program creation.

The purpose of the second time study was to determine, whether time savings could be achieved by annotating the 3D-models instead of creating 2D-drawings. The results of the study were, that the PMI-annotation approach actually takes slightly more time. Considering the issue a bit further, the result is quite natural, because basically the same annotation work was made with tools that are not familiar to the designers. If the tools would have been more familiar, the time consumed would have most likely been the same or slightly shorter as with the 2D-drawings. It was estimated by the test users that whether the design approach would be changed a bit and the dimensions from the sketches and modeling features would be applied straight to PMI, some amount of time would have most likely been saved.

The 3D-models can currently be utilized in almost any manufacturing process. As an exception, the sheet metal manufacturing is performed by utilizing the cutting DXFs. It is often said that the PMI-annotations could be utilized in the creation of a CAM-program. The annotations would be especially beneficial if they could be automatically utilized by a CAM-software. However, problems might occur if neutral formats are utilized and certain data is lost in the format exchange. The same issues most likely apply to the creation of the CMM-programs as well.

Based on the user experiences, the implementation of the MBD-environment should be made gradually with small steps. A way to begin the process is to execute a pilot project concerning the issue and including the most suitable persons and departments to the project. An important thing is that the possibility to still create the 2D-drawings should be maintained, because the usage of them cannot be terminated immediately.

6 Conclusions & Future Work

6.1 Summary

The current trends indicate that the manufacturing processes are developing towards more automated and digitalized environment. Naturally, the intelligent manufacturing systems require intelligent models to be performed as efficiently as possible. To make the digitalized manufacturing possible, the quality of the 3D-models must be high. The high quality design and efficient manufacturing are great possibilities for the western nations including Finland to compete against the developing markets.

An approach, that has been claimed to increase the productivity of the design and the manufacturing, is MBD. The benefits reported by several sources include:

- Decreased design time
- Reuse of the PMI-data in other applications (e.g. CAM)

In addition, several benefits that can be considered as facts, will be achieved. Two examples of the benefits are listed below:

- Single unambiguous source of the data
- Better visualization of the part

Konecranes utilizes mainly NX as a CAD-software. The annotated 3D-models can be created by NX, which creates a lightweight JT-file of each 3D-model automatically. The JT-file can be utilized in the collaboration between the mechanical design and the manufacturing and no additional investments are required. To manage the JT-files without NX, a free viewer (JT2Go or XpresReview) must be downloaded. The manufacturing departments investigated within the thesis are reasonably well capable of utilizing the 3D-models. However, the wide scale of different systems among the manufacturers might cause certain issues in the compatibility. Hence, the utilization of the STEP-file as a carrier of the PMI is going to be an important issue for the future investigations.

As a part of the thesis, the practical implementation of MBD within the CAD-software was investigated. The universal approach is that the PMI-annotations are applied to certain planes similarly as they are applied to the different projections in a 2D-drawing. The annotations are also typically displayed only in the view in which the annotations are best visible. In NX, the PMI-annotations can be applied manually or by utilizing already defined dimensions of the sketches and features of the 3D-model. The MBD-environments of the two CAD-software, NX and SolidWorks, were investigated. They have a slightly different approach to the way the collaboration between the design and manufacturing departments should be conducted. SolidWorks creates a 3D-PDF, which is almost like a 2D-drawing with views that can be rotated and zoomed. The JT-file created by NX resembles more the original 3D-model and can be utilized in the other Siemens software such as NX CAM.

There are several different file formats related to MBD. The formats JT and 3D-PDF are used for the visualization purpose, whereas the formats STEP and IGES can be used in other operations, including CAM-programming. The features defined in the standard ISO 16792, including highlighting the associated features and managing the visibility of the PMI-annotations, apply only to the visualization formats. The PMI-annotations from the original NX-model transfer quite well into the STEP-file, but the association with the features is not possible after the export. Hence, the reuse possibility of the PMI of a STEP-model in the CAM-programming is unlikely.

According to the time studies conducted during the thesis, the annotation work takes about the same time as the modeling of the geometry. The modeling section in the studies did not include the preliminary sketching of the geometry. The dimensioning of only the main dimensions naturally reduces the time needed for the annotating. No time savings were achieved by applying the annotations to the 3D-model instead of the 2D-drawing. More experience in the use of the PMI-tools and better utilization of the dimensions of the sketches and modeling features would most likely reduce the time spent in annotating the 3D-model.

Based on the results of the design item survey, the most typical manufacturing methods among the Konecranes products are sheet metal manufacturing and machining. Hence, the 3D-model utilization in both of the processes was investigated more in details. In addition, the creation of the casting models was investigated. The CAM-programs for milling and turning can be created only from a 3D-model. The best practice would be that the CAM-software would automatically utilize the PMI-annotations in the program creation. Another possibility is that the programmer sees the annotations on the model and based on these, applies certain operations manually. The sheet metal manufacturing utilizes mainly the 2D cutting DXF-files in the CAM-program creation. The 3D-models can typically be utilized only in the visualization. The creation of the casting models was done from a 3D-model only. No annotations were needed in the creation, meaning that the current 3D-models with no PMI-annotations suit the method already.

6.2 Discussion

MBD seems to be the direction in which the industries are heading to. However, the metal industry might be quite conservative in their working methods. Sometimes the conservativeness may be reasonable, because being a pioneer may be risky and expensive. However, in this case the risks are very small, because the implementation of MBD does not require any additional investments and the 2D-drawings can still be produced if needed. In addition, the manufacturing organizations already have rather developed systems that can utilize the intelligent data.

There are several undisputable benefits achieved by the use of MBD. The 3D-visualization will be a necessary feature in the future as the understanding of the 2D-drawings is getting weaker. After 10 to 20 years, a major amount of the designers and other professionals will not have a proper understanding of the 2D-geometries. Additionally, the products may be get more complex, which intensifies the need for a good visualization. An additional benefit is that the drawing files require a lot of space, which would be freed in an MBD-environment. Even though the data space is not expensive, it is always easier to handle a smaller amount of data in certain cases such as updating of the systems.

Certain often mentioned MBD-benefits are not as obvious as the previous. One of the most interesting issues is the time spent in the design phase. Even though there were no time savings achieved during the time studies of the thesis, it is probable that certain savings could be achieved. An important issue is to familiarize the designers with the PMI-tools and utilizing the already existing model dimensions as much as possible. As an example, several roundings of the same size can be applied to the same feature, and all of them can be dimensioned by making a single feature dimension visible. By selecting the annotation, all the associated roundings will be highlighted around the model.

Several indirect time savings and benefits could most likely be achieved as well. As an example, the prototype manufacturing could be made with a trusted manufacturer by using a 3D-model with only basic dimensions. In certain cases, even the production model could include only the main dimensions (e.g. sheet metal manufacturing and manufacturing of casting models). The second indirect time saving would be that the time spent in the fixing of the ambiguities and explaining the 2D-drawings to the manufacturer would certainly decrease.

A typical benefit that is always mentioned in the MBD-literature is the single data source. However, it is not fully realized when utilizing a JT or a 3D-PDF for the visualization, and an external STEP- or IGES-file for the creation of the CAM-program. In Anark Core systems the STEP-files can be embedded to the 3D-PDF, which is a good approach if the models are updated in sync. Embedding a STEP-file into a JT would also be a beneficial solution within Siemens products. An even better solution would be, if the annotated JT-file could be utilized within all the CAM-software among the manufacturers.

Although the JT-files cannot currently be opened with other CAD-software than NX, JT is a reasonable format to do the visualization. It is a lightweight format, which can be created by the current systems. In addition, it can be visualized in the standardized way with easy to use, extensive and free viewers. Another reasonable format is STEP. The versions 203 and 214, which can be exported from the NX 8.5 are typically usable by almost any software. In addition, the PMI-annotations transfer to the models, even though the association to features does not remain after the export. The new file format STEP 242 "Managed model based 3D engineering" is an interesting option for the data exchange. It can be assumed that the STEP 242 would include improvements to the exchange of the PMI-annotations among various CAD- and CAM-software. The STEP 242 will be available in NX 11, which will be published in June 2016.

In the beginning of the thesis four research questions were defined:

- Can model-based definition be implemented in current design and manufacturing environment?
- How can the data transfer be implemented if there is no longer 2D-drawings?
- How can the annotated 3D-models be utilized in different stages of the process?
- Can model-based definition save time and increase quality during the overall process?

As an answer to the first question, the mechanical design can easily be done in the MBD-environment. The current version of NX has the capability to produce the annotated 3D-models. In addition, the manufacturing departments are fairly well equipped for utilizing the 3D-models and intelligent data. Any manufacturer can also download a no-cost JT-viewer and gain the benefits of the better visualization without any other capability to utilize the 3D-models. The best answer to the second question considering the current environment is JT. The JT-file can be created with the current systems and the visualization follows the standard ISO 16792. 3D-PDF would also be a reasonable option, but to be able to create them, additional investments would be needed.

There are several answers to the third question. The first utilization possibility is the visualization of the part geometry and annotations. The second possibility is the CAM-programming, which can be done based on the 3D-models within most of the manufacturers.

One issue about the utilization of the 3D-models in the CAM-programming is that the PMI-annotations are no longer linked to the features, if using neutral formats are used. The answer for the last research question is that time saving could not be proved within the time studies. However, several direct and indirect time savings are possible, if the PMI-tools are be familiar to the designers and the model dimensions can more efficiently be utilized as the PMI-annotations.

6.3 Recommendations for the future

As a result of the thesis, certain recommended actions for the future are stated. The actions are based on the own observations and on the step-by-step guide by Boucher (2015). MBD should be taken into use in a pilot product development project, which should be small enough to be kept easily under control. In addition, suitable designers and a trusted manufacturer should be involved in the project to ensure a positive attitude towards a new method. The feedback and general experiences of the project should be gathered and certain actions based on them should be performed. In addition, the feedback and experiences can indicate whether it is beneficial to implement the method to other projects, or rather wait for the technologies to develop further. In any case, the evolution of the MBD-technologies should be followed in the future. The implementation of MBD has a low risk and a high possible reward. The 2D-drawings can still be easily taken back to use, if the MBD-tools are not developed enough for the current requirements.

Some additional studies are recommended to gather more knowledge about MBD. The first additional research topic is the utilization of the PMI-annotations within the format STEP. Investigations on how the PMI-annotations of a STEP-file can be utilized within different applications should be performed. In addition, the STEP 242 should be investigated further when the possibility to create them arises. The JT-file should also be taken under further investigations. Currently, JT-file can basically be utilized for the visualization purpose only. An ideal situation would be that the same JT-file could be used for both visualization and the creation of the CAM-programs. Another possibility would be to embed a STEP-file to the JT-file to make them update in sync. In addition to the file format researches, a new time study should be performed after the PMI-tools are more familiar to the designers. It is probable, that certain time savings could be achieved this way.

References

AIA; ASD; PDS, Inc; ProSTEP iViP, 2009. *Development of a Convergent Modular STEP Application Protocol Based on AP 203 and AP 214: STEP AP 242 – Managed Model Based 3D Engineering,* s.l.: STEP AP 242 Project.

Anark, 2016. *Anark Core FAQ.* [Online]
Available at: http://www.anark.com/core/core-faq.aspx
[Accessed 17 2 2016].

Autodesk, 1998. *DXF reference.* [Online]
Available at: http://www.autodesk.com/techpubs/autocad/dxf/reference/
[Accessed 16 12 2015].

Bhandankar, M., Downie, B., Hardwick, M. & Nagi, R., 2000. Migrating from IGES to STEP: one to one translation of IGES drawing to STEP drafting data. *Computer in Industry,* 41(3), pp. 261-277.

Black, R., 2002. *Design and manufacture : an integrated approach.* 1 ed. Basingstoke: Palgrave Macmillan.

Boothroyd, G., P, D. & Knight, W., 2011. *Product Design for Manufacture and Assembly.* 3 ed. Boca Raton, Florida: Taylor & Francis.

Boucher, M., 2015. *Tech-Clarity Insight: The How-to Guide for Implementing PMI.* Tech-Clarity.

Camba, J., Company, P., Contero, M. & Otey, J., 2014. *A review of the design intent concept in the context of CAD model quality metrics,* Indianapolis: 121st ASEE Annual Conference & Exposition.

Catalyst Connection, 2012. *Assessing the use of PDF based 3D Technical Data Packages within the defense industrial base.* [Online]
Available at: http://model-based-enterprise.org/Docs/ARDEC-3DTDP-assessment-findings.pdf
[Accessed 17 11 2015].

Claesson, O., 2015. *PLM Group - PLM Webinars - How to add PMI on your 3D CAD model.* [Online]
Available at: https://plmgroup.fi/webinaari/how-to-add-pmi-on-your-3d-cad-model/
[Accessed 18 1 2016].

Feeney, A. & Frechette, S., 2002. *TESTING STEP-NC IMPLEMENTATIONS.* [Online]
Available at: http://www.mel.nist.gov/msidlibrary/doc/WAC20022.pdf
[Accessed 11 12 2015].

Feeney, A. & Hedberg, T., 2014. *STEP: Standard for the Exchange of Product model data.* [Online]
Available at: http://www.nist.gov/el/msid/upload/16_aBarnardFeeney.pdf
[Accessed 11 12 2015].

Frechette, S., Jones, A. & Fischer, B., 2013. Strategy for testing conformance to geometric dimensioning & Tolerancing Standards. *Procedia CIRP,* Volume 10, pp. 211-215.

Gao, W. et al., 2015. The status, challenges, and future of additive manufacturing in engineering. *Computer-Aided Design,* 69(12), pp. 65-89.

Hasari, H. & Salonen, P., 2006. *Teknillinen piirtäminen.* 1 ed. Helsinki: Otava .

Heinonen, M. & Marjosalmi, M., 2012. *Ohjelmointityökalun päivittäminen - Konecranes Oy,* Espoo: Teknillinen korkeakoulu, Konetekniikan osasto, Tuotantotekniikan laboratorio.

Hokkanen, M., 2015. *NC-programmer* [Interview] (13 10 2015).

Ihalainen, E., Aaltonen, K., Aromäki, M. & Sihvonen, P., 1995. *Valmistustekniikka.* 5 ed. Espoo: Otatieto.

Keating, D., 2012. *JT Powering the World of PLM.* Gaithersburg, Maryland, NIST MBE Conference 2012.

Kemmerer, S., 1999. *STEP The Grand Experience.* [Online]
Available at: http://www.mel.nist.gov/msidlibrary/doc/stepbook.pdf
[Accessed 15 12 2015].

Konecranes, 2011. *Konecranes Glossary.* [Online]
[Accessed 14 10 2015].

Konecranes, 2016. *General Description.* [Online]
Available at: http://www.konecranes.com/about/general-description
[Accessed 16 2 2016].

Kuutela, K. & Manninen, P., 2015. *Solidimallit CAM-ohjelmointiin.*

Lalit, K., Mallikarjuna, K. & Sarcar, M., 2008. *Computer Aided Design and Manufacturing.* 1 ed. New Delhi: Prentice-Hall.

Laukkanen, N., 2015. *Konecranes Product Development – Teamcenter/NX Tips.*

Lubell, J., Chen, K., Horst, J. & Frechette, S. H. P., 2012. *Model Based Enterprise / Technical Data Package,* National Institute of Standards and Technology.

Lubell, J. et al., 2013. *Model-Based Enterprise - Summit report,* s.l.: NIST.

Majerik, J. & Jambor, J., 2014. Computer Aided Design and Manufacturing Evaluation of Milling Cutter when High Speed Machining of Hardened Steels. *Procedia Engineering,* Volume 100, pp. 450-459.

Manninen, J., 2015. *Piirustuksetonta valmistusta - (osa 1).* [Online]
Available at: https://plmgroup.fi/webinaari/piirustuksetonta-valmistusta-osa-1/
[Accessed 23 11 2015].

Matilainen, J. et al., 2011. *Ohutlevytuotteiden suunnittelijan käsikirja.* 1 ed. Helsinki: Teknologiateollisuus.

MIL-STD-31000A, 2013. *Department of Defense Standard Practice: Technical Data Packages.*

Model Based Enterprise, 2014. *MIL Standards.* [Online]
Available at: http://model-based-enterprise.org/mil-standards.html
[Accessed 10 2 2016].

Mäkelä, T., 2016. *Interview of PMC Factory Manager* [Interview] (27 1 2016).

NIST MEP, 2009. *Assessment of Supplier Capabilities to Operate in a Model-Based Enterprise Environment,* NIST MEP.

Opsahl, D., 2013. *Positioning 3DPDF in Manufacturing.* [Online]
Available at: http://www.3dpdfconsortium.org/wp-content/uploads/2015/08/positioning-3dpdf-in-manufacturing_v1.pdf
[Accessed 9 12 2015].

Pere, A., 2004. *Koneenpiirustus 1 & 2.* 9 ed. Espoo: Kirpe.

Pikkarainen, E. & Mustonen, M., 2010. *Numeerisesti ohjatut työstökoneet.* 2 ed. Kajaani: Opetushallitus.

Popkov, A., 2014. *Harmonization of CAD tools in PLM environment,* Espoo, Finland: Aalto university.

Popkov, A., 2016. *PMI Training.*

Quintana, V. et al., 2010. Will Model-based Definition replace engineering drawings throughout the product lifecycle? A global perspective from aerospace industry. *Computers in Industry,* 61(5), pp. 497-508.

Radhakrishnan, P., Subramanyan, S. & Raju, V., 2008. *CAD/CAM/CIM.* 3 ed. New Delhi: New age international.

Rapinoja, J.-P., 2015. 2d-piirustukset jäävät historiaan. *Metallitekniikka,* Issue 7-8, pp. 49-53.

Rosenqvist, J., 2015. *PLM Group - PLM Webinars - A Roadmap for Successfully Adopting Model-Based Enterprise.* [Online]
Available at: https://plmgroup.fi/webinaari/a-roadmap-for-adopting-model-based-enterprise-part-3/
[Accessed 17 2 2016].

SAAB, 2015. *Business areas.* [Online]
Available at: http://saabgroup.com/about-company/organization/business-areas/
[Accessed 25 11 2015].

SFS-ISO 16792, 2010. *Technical product documentation. Digital product definition data practices.* Helsinki: Suomen Standardoimisliitto.

Sicking, P., 2014. *JT - The 3D Standard for Today and Tomorrow.* Tokyo, Japan, Siemens PLM Software.

Siemens PLM Software, 2008. *Parasolid XT Format Reference.* [Online]
Available at:
http://www.plm.automation.siemens.com/en_us/Images/XT_Format_April_2008_tcm1023-62642.pdf
[Accessed 15 12 2015].

Siemens PLM Software, 2012. *Siemens' JT Data Format Accepted as the World's First ISO International Standard for Viewing and Sharing Lightweight 3D Product Information.* [Online]
Available at:
http://www.plm.automation.siemens.com/en_us/about_us/newsroom/press/press_release.cfm?Component=205727&ComponentTemplate=822
[Accessed 8 12 2015].

Siemense PLM Software, 2013. *Delivering quality components: Parasolid's drive for success.* [Online]
Available at:
http://www.plm.automation.siemens.com/en_us/products/open/parasolid/xt_pipeline/index.shtml#lightview%26uri=tcm:1023-50201%26title=Delivering-Quality-Components-White-Paper-9340%26docType=pdf
[Accessed 15 12 2015].

Simons, J., 2016. *Product & Manufacturing Information (PMI).* Ideal PLM.

SolidWorks, 2015. *SolidCAM Inc.: SolidCAM.* [Online]
Available at: https://www.solidworks.com/sw/products/details.htm?productID=69
[Accessed 18 12 2015].

SolidWorks, 2015. *WHAT'S NEW - SOLIDWORKS 2015.* [Online]
Available at:
http://files.solidworks.com/Supportfiles/Whats_new/2015/English/whatsnew.pdf
[Accessed 18 1 2016].

Spreier, P., 2013. *7 Minutes with a PDF standard.* [Online]
Available at: http://www.3dpdfconsortium.org/wp-content/uploads/2015/08/7-minutes-with-prc.pdf
[Accessed 9 12 2015].

Supplier A, 2015. *E-mail from sales manager.*

Supplier B, 2015. *E-mail from company representative.*

Supplier C, 2015. *E-mail from project manager.*

Supplier D, 2015. *E-mail from company representative.*

Supplier E, 2015. *E-mail from company representative.*

Supplier F, 2015. *E-mail from company representative.*

Whittenburg, M. & Whittenburg, R., 2013. *MIL-STD-31000A Putting The Pieces Together.* [Online]

Available at: http://model-based-enterprise.org/Docs/2-21-14%20C3%20Presentation%20MIL-STD-31000_A_Training_WH-1.pdf
[Accessed 20 11 2015].